PRACTICAL MANUAL ON ELECTRONICS

FOR VOCATIONAL COURSE AT THE HIGHER SECONDARY STAGE

VOLUME I

राष्ट्रीय शैक्षिक अनुसंधान और प्रशिक्षण परिषद्
NATIONAL COUNCIL OF EDUCATIONAL RESEARCH AND TRAINING

Practical Manual
on
Electronics

For Vocational Course at the Higher Secondary Stage

Volume I

N. P. BHATTACHARYA
Project Coordinator

विद्यया ऽमृतमश्नुते
एन सी ई आर टी
NCERT

राष्ट्रीय शैक्षिक अनुसंधान और प्रशिक्षण परिषद्
NATIONAL COUNCIL OF EDUCATIONAL RESEARCH AND TRAINING

February 1993
Magha 1914

P.D. 1T-VK

Publication Team

C. N. Rao *Head*, Publication Department

Prabhakar Dwivedi *Chief Editor*
Pooran Mal *Editor*
Vijay Kumar *Assistant Editor*

U. Prabhakar Rao *Chief Production Officer*
Shiv Kumar *Production Officer*
Subodh Srivastava *Production Assistant*
C. P. Tandan *Art Officer*

Cover C. P. Tandan

OFFICES OF THE PUBLICATION DEPARTMENT, NCERT

NCERT Campus	CWC Campus	Navjivan Trust Building	CWC Campus
Sri Aurobindo Marg	Chitlapakkam, Chromepet	P.O. Navjivan	32, B.T. Road, Sukchar
NEW DELHI 110016	**MADRAS 600064**	**AHMEDABAD 380014**	**24 PARGANAS 743179**

Rs. 25.00

Published at the Publication Department by the Secretary, National Council of Educational Research and Training, Sri Aurobindo Marg, New Delhi 110016, laser typeset at Compugraphics, H-16, Green Park Extn., New Delhi 110016 and printed at J.K. Offset Printers, 315, Jama Masjid, Delhi 110006

FOREWORD

The programme of vocationalization of higher secondary education has been accepted by the country as it holds forth great promise for linking education with productivity and economic development by providing education for better employability of the youth.

In view of the importance of the programme, the NCERT is making an all-out effort to provide academic support to the implementing agencies in the States. One of the major contributions of the NCERT is in the field of curriculum development and in the development of model instructional materials. The materials are developed through workshops in which experts, subject specialists, employers' representatives, curriculum framers and teachers of the vocational courses are involved.

The present *Manual on Practical Electronics* has been developed in the manner described above is meant for the students studying electronics course at the higher secondary stage. It is being published for wider dissemination amongst students and teachers throughout the country. I hope that they will find the manual useful.

I am grateful to all those who have contributed to the development of this manual. I also acknowledge the great interest taken by Prof. A. K. Mishra, Head of our Department of Vocationalization of Education in inspiring his colleagues in their endeavours to develop instructional material. My compliments are due to Shri N.P. Bhattacharya, Reader, who functioned as the Project Coordinator for the development of this manual.

Suggestions for improvement of this manual will be welcome.

K. GOPALAN
Director
National Council of
Educational Research and Training

PREFACE

Ever since the introduction of vocationalization of education in our school system by several States and Union Territories in our country, the paucity of the appropriate instructional materials has been felt as one of the major constraints in the implementation of the programme and source of great hardship to pupils offering vocational studies at senior secondary stage.

The Department of Vocationalization of Education of the National Council of Educational Research and Training, New Delhi, has started a modest programme of developing instructional material of diverse types to fill this void in all major areas of vocational education. The task is too gigantic to be completed by any single agency but the model material being developed by us might provide guidance and impetus to the authors and agencies desiring to contribute in this area. These are based on the national guidelines developed by a working group of experts constituted by the NCERT.

The present book on *Practical Manual on Electronics* is meant for the pupils and teachers of Electronics Technology course being offered in a number of States and Union Territories. It is hoped that the users will find it immensely useful.

The material was developed during workshops held in the Workshop Department, NCERT, New Delhi. The name of the contributors are mentioned elsewhere and their contributions are admirably acknowledged. Shri N.P. Bhattacharya, Reader and Coordinator of the project, Department of Vocationalization of Education and Resource persons Shri N. N. Mohanty and Shri M. M. Kaushik deserve special thanks for bringing out the materials.

ARUN K. MISHRA
Professor and Head
Department of Vocationalization of Education

GANDHIJI'S TALISMAN

"I will give you a talisman. Whenever you are in doubt or when the self becomes too much with you, apply the following test :

Recall the face of the poorest and the weakest man whom you may have seen and ask yourself if the step you contemplate is going to be of any use to him. Will he gain anything by it ? Will it restore him to a control over his own life and destiny ? In other words, will it lead to Swaraj for the hungry and spiritually starving millions ?

Then you will find your doubts and your self melting away."

mkGandhi

CONTENTS

FOREWORD iii
PREFACE v
INTRODUCTION 1

Experiments

1. Semiconductor Diode 9
2. Resistance of a Diode 11
3. Half Wave Rectifier 14
4. Full Wave Rectifier 16
5. The Zener Diode 18
6. Bipolar Junction Transistor 22
7. Basic Action of a Transistor 25
8. Transistor as an Amplifier 27
9. Transistor as a Switch 30
10. Darlington Pair of Transistors 33
11. Audio Oscillator 35
12. RC Coupled Audio Amplifier 38
13. Thermistor 40
14. Light Dependent Resistor (LDR) 42
15. Field Effect Transistor 44
16. Field Effect Transistor (FET) as an Amplifier 48
17. Monostable Multivibrator 50
18. Bistable Multivibrator 53
19. Astable Multivibrator 56

Introduction of Experiments on Logic Gates 58

20. Logic OR - gate 59
21. Logic AND- gate 61
22. Logic NOT- gate 63
23. Logic NAND- gate 65
24. Logic NOR- gate 68

Practical Projects 71

25. Construction of a Dual-power Supply Unit 73
26. Construction of Call Bell Using a IC-741 76
27. Construction of a Timer Circuit Using IC-555 78

ANNEXURES *79*

I. Units and Symbols 79
II. Resistors 80
III. Capacitors 82

APPENDIX *84*

List of Contributors 84

Introduction

Learning is best done by doing a thing with your own hand. We have thus laid more emphasis on practicals in your vocational course on Electronics.

The first part of this "Practical Manual on Electronics" has been prepared to guide you to do simple experiments in your school laboratories with equipment and materials easily available. Also these experiments have been judiciously chosen, so as to make you thoroughly conversant with fundamental aspects of the subject. We have included three practical projects which you should construct and study as detailed therein. At the end of this book we have included three annexures for your ready reference and use. They are about standard units and symbols, resistors and capacitors.

Certain common pre-requisites are necessary for all the experiments described in this manual. For example, you will require certain basic minimum common facilities to conduct them.

Circuit Boards

A suitable circuit board like an "Experimental Bread-Board will be required (Fig.0.1). In this board, there will be spring-loaded contact points or junction points, where the ends of resistors, capacitors etc. can be joined together.

As an alternative, you may have an all purpose printed circuit boards (PCB) or strip boards. In such a case, the end points of the circuit components have to be soldered. You may also have a board with several points where screws have to be tightened to join several components. All these types of boards can easily be used to make the circuit.

Soldering

Even though the experiments will involve a minimum of soldering, as a student of the Vocational Course on Electronics, you have to be familiarised with techniques of soldering. Soldering by itself is a specialised art, but with a little bit of practice, the type of soldering you

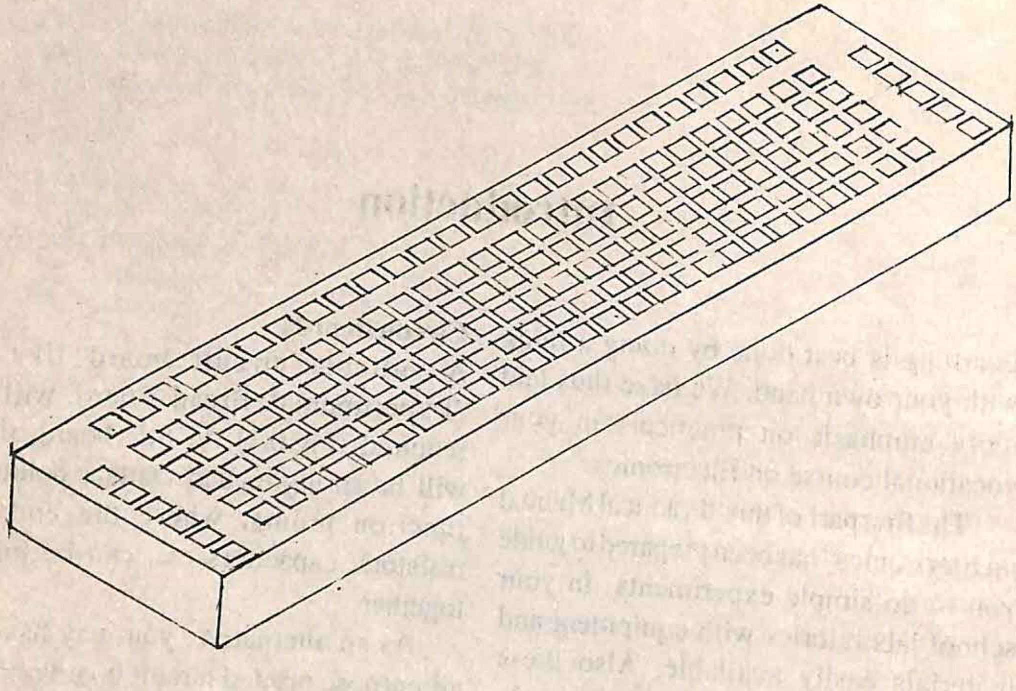


Fig. 0.1 Bread Board

are able to do is good enough for your experiments.

Needless to say, you have to use a good quality soldering iron, preferably around 35 Watts capacity for your experiments. Your soldering iron is a mains operated device. It works from 230 Volts A.C. supply commonly available in our labs. You have therefore to observe certain precautions, so as to protect yourself from electric-shock. It is a good practice to do your experiments using the soldering iron, while standing on an insulated surface like rubber mat. In other words, you must follow all safety precautions as per standard laboratory practice.

Your soldering iron gets heated up and you must avoid touching the hot portions, nor should you take the soldering iron anywhere near plastics or rubbers. After use, always leave it on the stand and inside its housing.

Quite often, even though you might have rigged up the circuit very well, you may get faulty results. This may be due to bad solder joints. You must use good quality solder material with resin core. The ends of the circuit components must be cleaned and tinned with the help of the

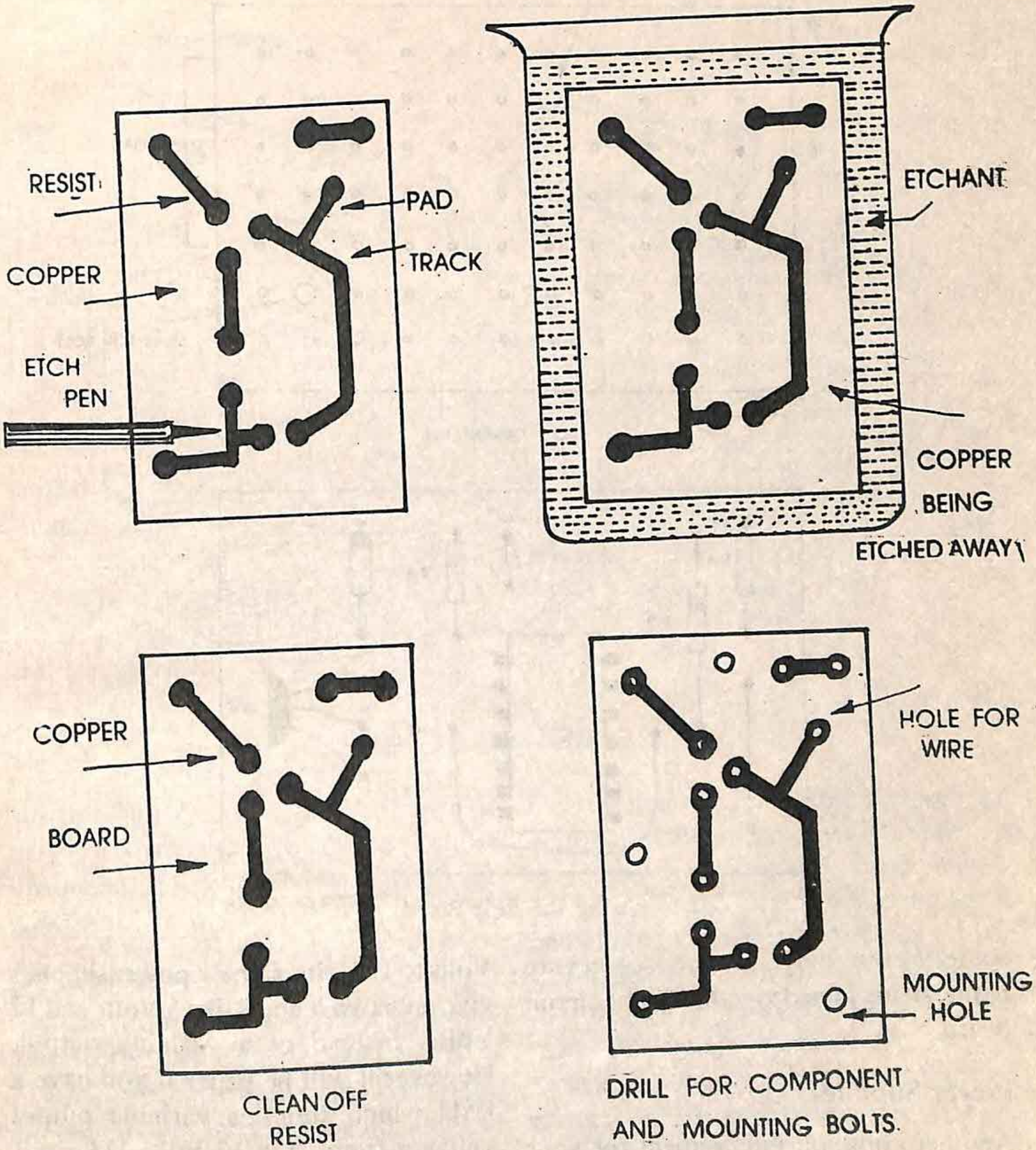


Fig. 0.2 Printed Circuit Board

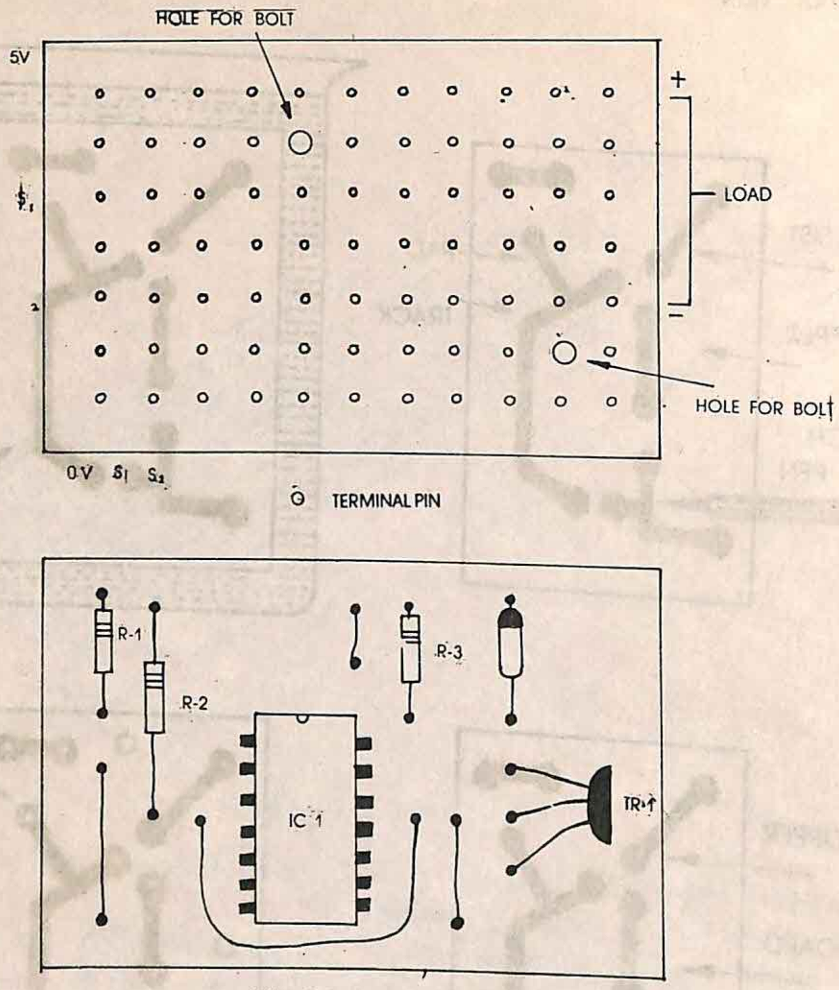


Fig. 0.3 Strip Board

soldering iron and resin-cored solder wire before being joined together on the circuit board.

Power Supplies

Another common requirement for your experiments will be a d.c. power source. The power supply unit (PSU) will operate from a 230 Volts A.C. source and at its output it will give a variable d.c. of 5 Volts to 12 Volts. Certain power supplies give only two outputs like 5 Volts and 12 Volts, instead of a variable output. However it will be better if you have a PSU which supplies variable output voltages from 0 to 12 Volts D.C. at 1 ampere.

Instead, you may use dry cells like torch batteries or other type as commonly available in the market. You have to use

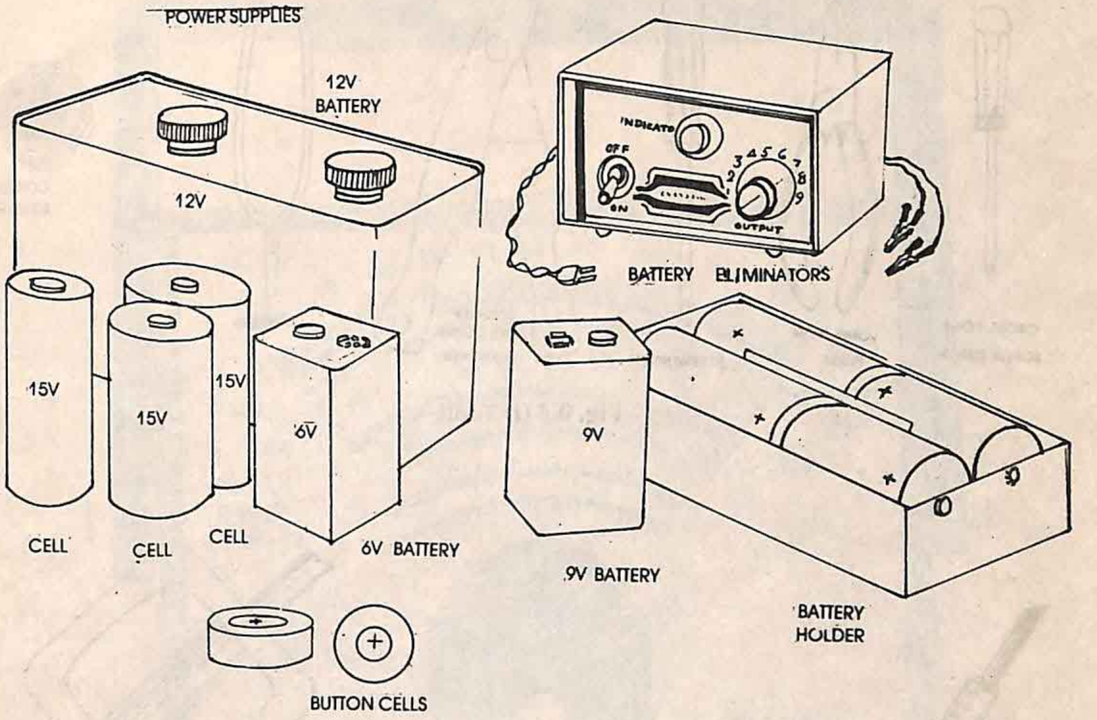


Fig. 0.4 Power Supplies

suitable battery boxes, clip terminals or screw terminals to take out the positive and negative leads as shown in the figure given.

Tools and Multimeter

Common tools like, screw drivers, cutting pliers, nose pliers, wire stripper, tweezers etc. as commonly used in the lab, would be required for the experiments. Figures for such common tools are given.

An indispensable instrument for all your experiment would be a common type of multimeter. It need not be a very specialised one. It should be good one to measure resistance, current and voltage with reasonable accuracy without loading the circuit. It usually has special leads to connect it to the circuit. At one end, it has plugs to connect to sockets on the meter. At the other end they have needle like probes or crocodile clips. Before using the meter you must take care to switch it to the correct voltage or current range. While measuring resistance, switch to a range that includes the expected value of the resistance. Make sure that you are not touching the bare metallic portion of the probes with your fingers while using the multimeter.

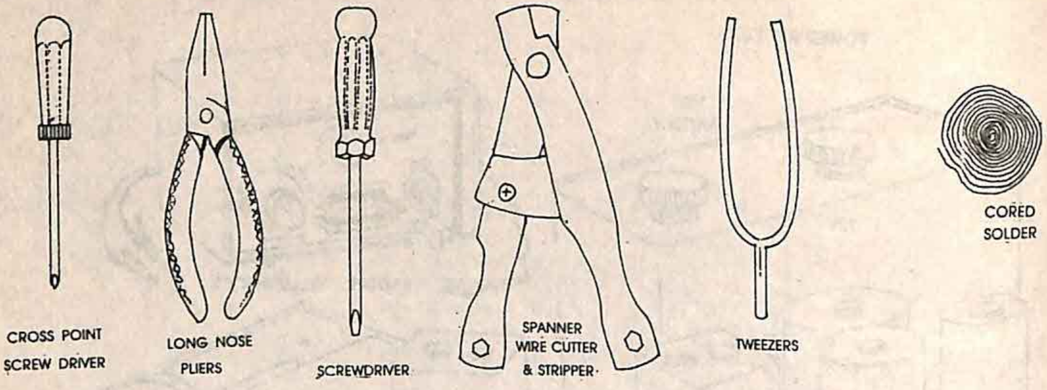


Fig. 0.5 (i) Tools

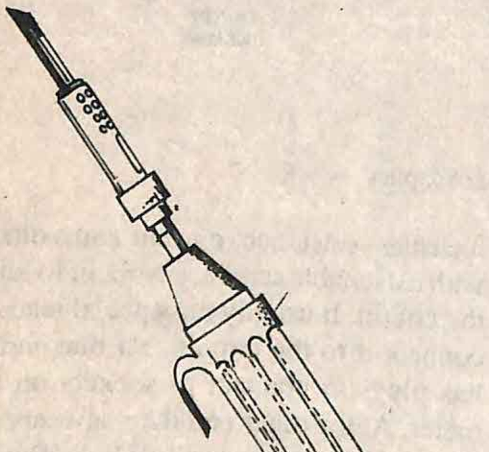


Fig. 0.5 (ii) Tools

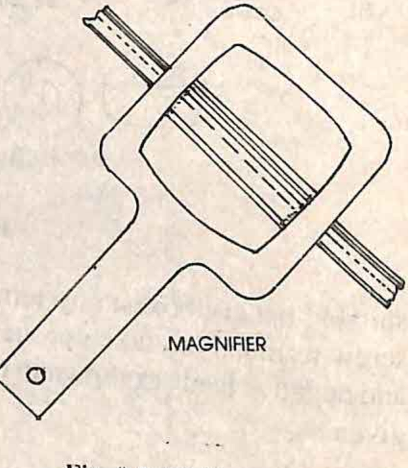


Fig. 0.5 (iii) Tools

Experiments

In case of each of your experiments, we have written out the Objective, so that you know, what exactly you will learn after conducting this experiment. Instead of loading you up with too much of a theory, we have very briefly and in a

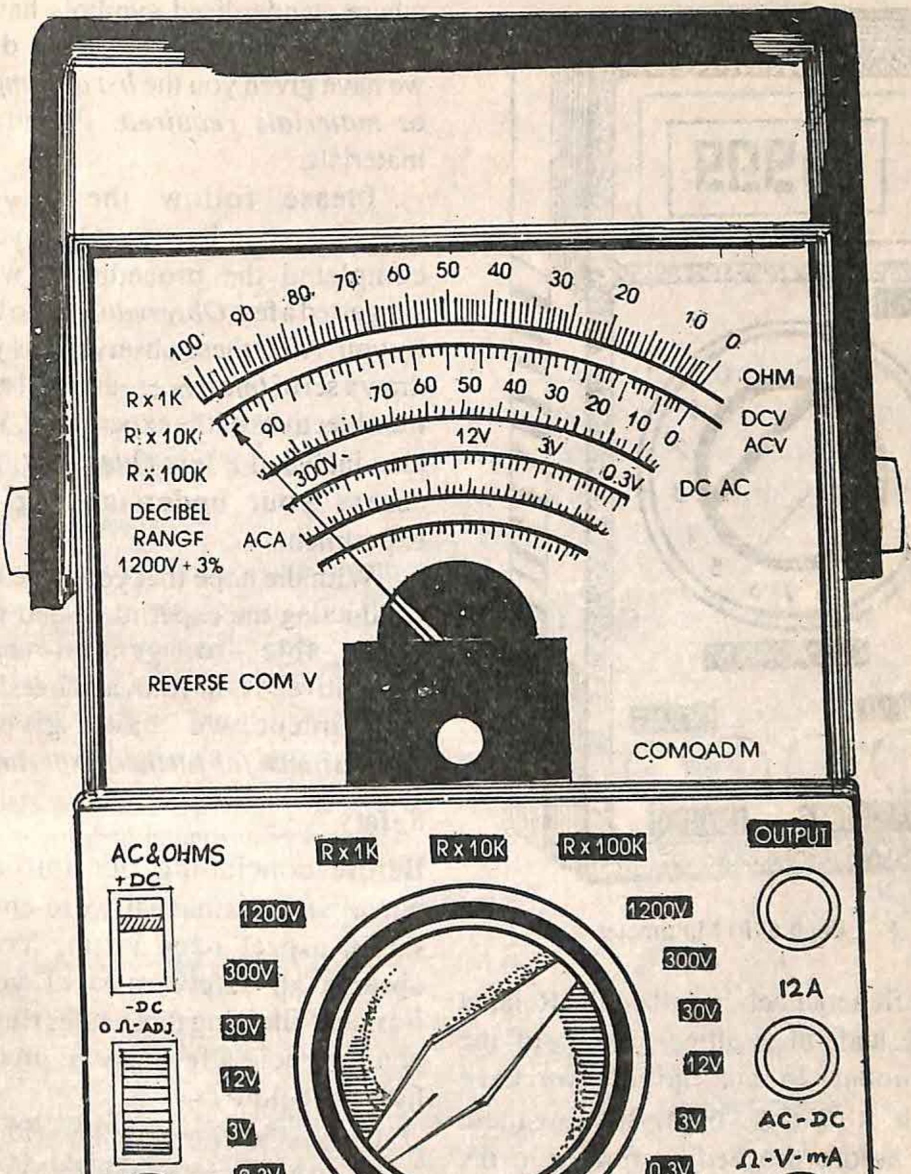


Fig 0.6 (i) Multimeter

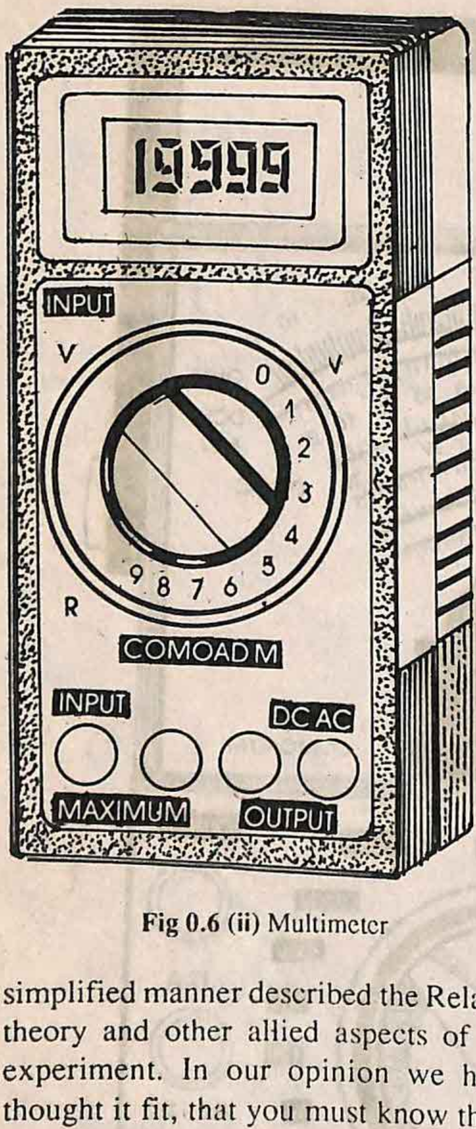


Fig 0.6 (ii) Multimeter

simplified manner described the Related theory and other allied aspects of the experiment. In our opinion we have thought it fit, that you must know these facts, before proceeding further in the experiment.

Following this, we have drawn the *Circuit diagram* for the experiments. You must get practised in studying such simple circuit diagrams, where normally the flow of events is from left to right, and where standardised symbols have been utilised. Following the circuit diagram, we have given you the *list of components* or *materials required*. Procure these materials.

Please follow the *Procedure* thereafter, step by step. Once you have completed the procedures, we have suggested a few *Observations* to be made by you. After these observations you will draw a set of *Inferences*, that will establish the objectives of the experiment. We have also included a few *Questions* so as to assess your understanding of the experiment.

With the hope that you have enjoyed conducting the experiment and we have been able to generate certain inquisitiveness in your, at the end of each experiment we have given you *Suggestions for further Experiments*.

Safety

Before concluding our introductory remarks for this manual, we re-emphasise safety aspect once again. You must observe all safety rules of your lab. Besides following those rules rigorously, you may note a few special precautions listed as below :—

(i) Avoid direct contact with voltage sources, specially 230 Volts A.C. mains.

(ii) Do not connect any circuit components without switching-off the power-supply.

(iii) Replace damaged plugs and cables.

1. Semiconductor Diode

Objective

To show that a p-n junction diode conducts in one direction only.

Related Information

A p-n junction diode is a two terminal device with a p-type and n-type semiconducting material. The p-side is called the "anode", and the n-side the "cathode". The most important characteristic of a p-n junction diode is its ability to conduct current in one direction only. It's symbol is shown in Fig. 1.1.

Fig. 1.1

In actual diodes the cathode end is marked by a band as shown in Fig. 1.2.

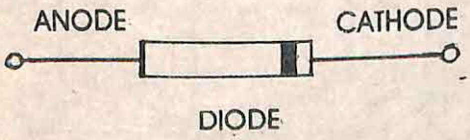


Fig. 1.2

The diode is said to be forward biased when the anode is made positive with respect to the cathode. The diode produces large current when forward biased. (Fig. 1.3)

Circuit Diagram

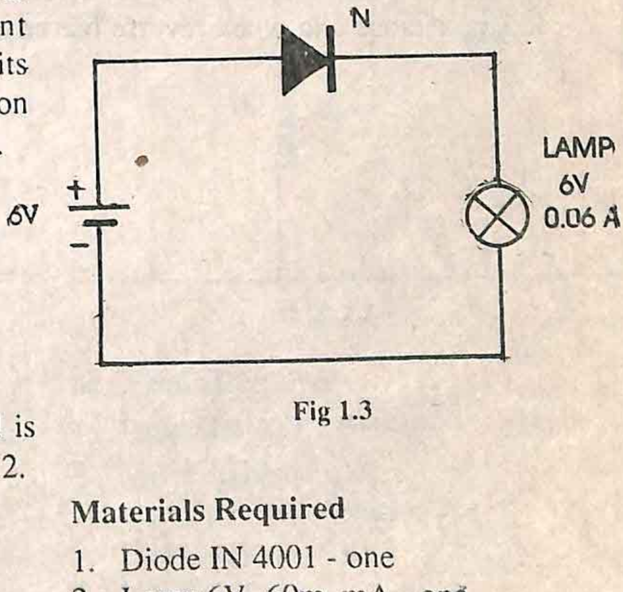


Fig 1.3

Materials Required

1. Diode IN 4001 - one
2. Lamp 6V- 60m mA - one
3. Power supply 6V- one
4. Connecting wires

Procedure

1. Make the connections as shown in Fig. 1.3.
2. The p-n junction is forward biased. Observe the lamp.
3. Now reverse the position of the diode so that now the anode is made negative with respect to the cathode. This is known as reverse biasing of the diode. Now observe the lamp.

Observations

When the diode is forward biased the lamp glows brightly, however when the diode is reverse biased the lamp does not glow.

Inference

A diode when forward biased offers low resistance and when reverse biased offers high resistance.

Questions

1. When is the diode forward biased ?
2. When is the diode reverse biased ?

Suggestion for Further Experiments

An LED with a resistance in series can be used in place of the lamp for carrying out this experiment. The circuit is shown in Fig. 1.4.

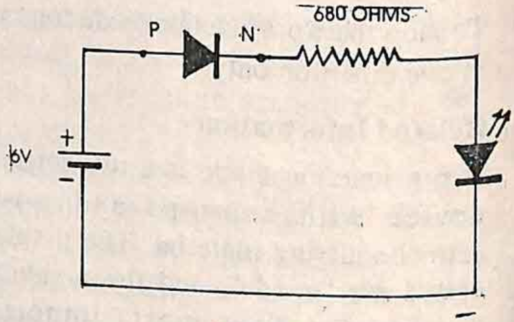


Fig. 1.4

2. Resistance of a Diode

Objective

To study the variation of current with voltage of a diode in the forward as well as reverse bias condition.

Related Information

The diode when forward biased conducts and when reverse biased it does not conducts. Thus, a diode when forward biased offers low resistance and when reverse biased offers high resistance to the flow of current.

The 1k potentiometer in the Fig. 2.1 acts as a voltage divider. At various settings of the potentiometer the output voltage varies. When we apply a voltage of $V_i = 6V$, as in Fig. 2.1 we can get voltages $V_o = 0.1V, 0.2V, 0.3V$ etc. by varying the potentiometer.

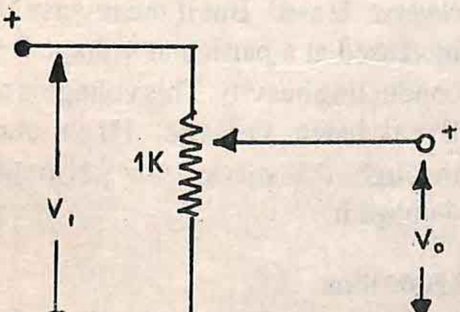


Fig. 2.1

Circuit Diagram

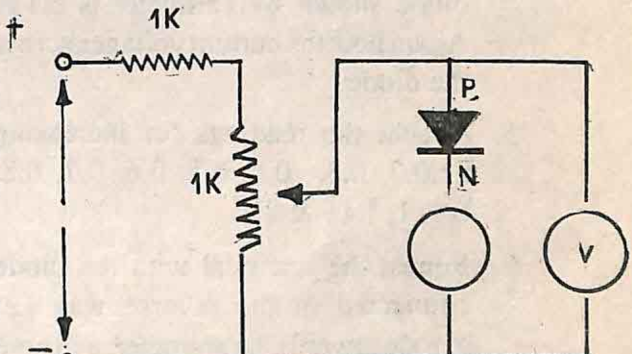


Fig. 2.2

Materials Required

1. Diode IN 4001- One
2. Variable Resistance (1k Ω potentiometer) - One
3. Power supply (6V)
4. Voltmeter
5. Ammeter
6. Resistance 1/4 One Watt One (*Note* : Instead of the voltmeter and

ammeter, a multimeter can be used to make the measurements.)

7. Switch—One.

Procedure

1. Make the connections as shown in Fig. 2.2, but do not connect the diode.
2. Switch on the power turn the potentiometer fully anticlockwise so that the voltage at the variable point of potentiometer is 'O' Volts.
3. Now connect the diode as shown.
4. Turn the potentiometer knob slowly clockwise until the voltage across the diode shown by voltmeter is o.1V. Again note the current voltages across the diode.
5. Repeat the readings for increasing i.e.0.2, 0.3, 0.4, 0.5, 0.6, 0.7, 0.8, 0.9, 1, 1.15 & 2V.
6. Repeat the practical with the diode connected in the reverse way i.e. amode towards the ammeter and note the current through the diode.

Observations

The reading obtained for voltage and current should be recorded in a tabular form and then resistance can be calculated by the formula R = V/I. If voltages across the diodes are in volts and corresponding currents in milliamps then the resistance is given by

$$R = \frac{V}{I} \times 1000 \text{ ohms.}$$

Forward Bias Table

(V)	(I)	R
in volts	(in mA)	(in ohm)

Draw a graph showing how the current varies with voltage. Note the result when the diode was connected in reverse-biased condition.

Reverse Bias Table

(V)	(I)	R
(in volts)	(in mA)	(in ohm)

Inferences

When the voltages across the diode is increased the diodes start conducting and after a certain value of p.d's the current increases rapidly. The voltage where current starts to increase rapidly is called the "knee oı offset voltage." For a silicon diode, this voltage equals the barrier potential, around 0.7V. (The germanium diode has an offset voltage of 0.2V). The diode normally does not conduct when reverse biased. But if the reverse bias is increased at a particular voltage it starts conducting heavily. This voltage is called Breakdown voltage. High current through the diode can permanently damage it.

Questions

1. Does the resistance of the diode remain constant as the voltage across it is increased ?

2. At what range of voltages the diode has extremely high resistance ?

3. When the voltage across diode is increased, what happens to its resistance ?

Suggestions for Further Experiments

Repeat the experiment using a germanium diode OA47 or OA91. Compare the results with those you got for the silicon diodes.

3. Half Wave Rectifier

Objective

To derive d.c. voltage out of an a.c. voltage by rectifying only one half cycle of the a.c. waves by using a diode.

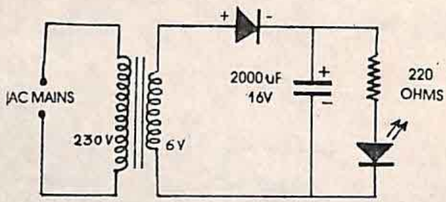


Fig. 3.1

Related Information

A diode conducts well in the forward direction and almost nothing in the reverse direction. The amount of current depends on the voltage. The conversion of alternating current into direct current is known as rectification. Since the diode conducts only in one direction so it is ideal for use as a rectifier. It conducts only during the positive half cycle of the a.c. wave. The diode does not conduct during the negative half cycle. A rectifier using one diode is therefore called a half wave rectifier.

Circuit Diagram

Materials Required

1. Diode IN 4001 - One
2. Electrolytic Capacitor 2000 MF/16V - One
3. LED - One
4. Resistance 220 ohm - One
5. Step down transformer 220 V/6V - One
6. Connecting wires etc.

Procedure

1. Make the connections as shown in Fig. 3.1 Observe the glow of the LED.
2. Reverse the orientation of the diode. Observe the LED.

3. Remove the capacitor and observe the LED.

Observations

When the diode is connected as shown in Fig. 3.1 the LED glows brightly. When the orientation of diode is reversed the LED does not glow. After the capacitor is removed the LED glows with less intensity than before.

Inference

LED does not glow when the orientation of diode is reversed showing that the lamp is supplied with unidirectional current (d.c.). When the diode is forward biased the LED glow only in one half cycle of a.c. and not in the other half cycle but it appears to glow continuously. In fact the LED switches off and on as the a.c. changes from positive to negative half cycle with naked eye however this change is not visible. The capacitor can be thought of as 'filter' which filters out a.c. and blocks d.c. so that almost pure d.c. is provided for the lamp and hence the bright light.

Questions

1. What is rectification ?
2. What is a half-wave rectifier ?
3. How will the intensity of lamp vary when the capacitor is removed ?

Suggestions for Further Experiments

If you have a cathode ray oscilloscope (C.R.O.) you can observe the a.c. waveforms of the supply line and d.c. output from the diode.

Make the connections as shown in Fig. 3.2.

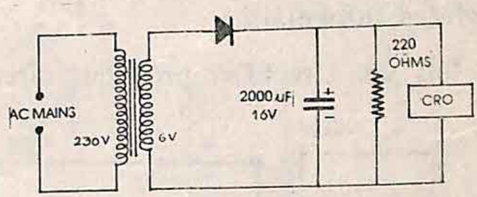


Fig. 3.2

Set up the C.R.O. Set the Y-amplifier control of C.R.O. to 10 V/cm and time base (Y) to 10 ms/cm. Observe the waveform. Again reverse the diode. Observe the waveform. Now remove the capacitor and observe the waveform.

4. Full Wave Rectifier

Objective

To show that four diodes forming a bridge network act as a full wave rectifier.

Related Information

A full wave rectifier produces direct current from both halves of the a-c waveform. Two diodes can be used to make a full wave rectifier. However, the most commonly used full wave rectifier has four diodes, connected in a bridge form as shown in the circuit diagram shown in Fig. 4.1.

Materials Required

1. Diode In 4001- Four
2. Electrolytic capacitor (4.7μF) - One
3. Resistance 220 ohms - One
4. LED - One
5. Step down transformer (230V:6V) - One
6. Connecting wires
7. Switch—One.

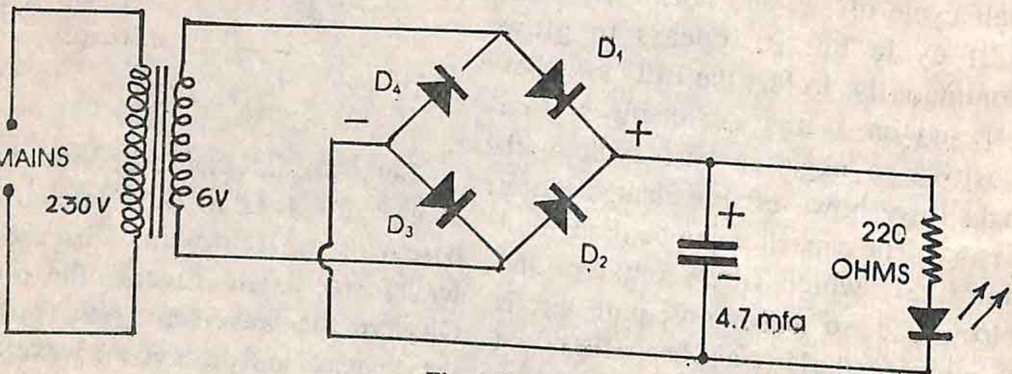


Fig. 4.1

Procedure

1. Make the circuit as shown in Fig. 4.1 and observe the glow of the lamp.
2. Remove any one of the diodes and

observe the glow of lamp.

Inference

In the first case when all diodes are connected as shown the full wave rectification is obtained lamp glows brightly where as in the 2nd case when a single diode is removed one half cycle is rectified and the lamp glows less brightly.

Questions

1. What do you mean by full wave rectification ?
2. When you decrease the resistance of the load, does not rectifier delivers larger current to the load ?
3. What is the function of the capacitor.

Suggestions for Further Experiments

If you have a C.R.O. this experiment can be performed to observe the waveforms. Remove the lamp or LED alongwith 200 ohms resistance and connect a 820 ohm resistor in its place. Make the circuit as shown in Fig. 4.2.

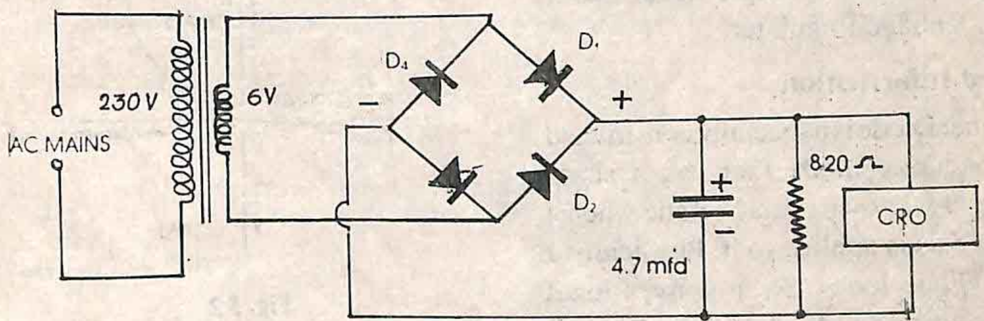


Fig. 4.2

Set up the C.R.O. with its Y-amplifier control to 10 V/cm. Set the time base 10ms/cm.
Switch on the supply and observe the waveform. Replace the capacitor with a 100 micro-farad capacitor. Observe the waveforms.

5. The Zener Diode

Objectives

1. To study the effects of forward and reverse bias on current in a Zener Diode.
2. To introduce a simple Zener Diode as Voltage Regulator.

Related Information

The Zener Diode is a specially constructed semiconductor diode that has a sharp voltage breakdown characteristic when a reverse bias is applied to it. Physically a Zener Diode looks like a conventional semiconductor diode. The circuit symbol of a zener diode is shown in Fig. 5.1

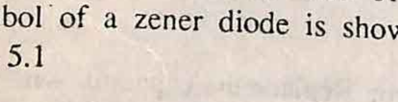


Fig. 5.1

When operated in its breakdown region, the voltage across it remains at a fixed value for a large range of current as shown in Fig. 5.2.

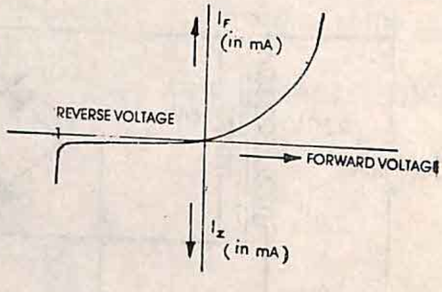


Fig. 5.2

For this reason, this type of diode can supply a fixed regulated voltage even when the supply voltage changes.

The rating of Zener Diode specify the Zener Voltage (V_z), tolerance range, Zener current limits, maximum power dissipation and maximum operating temperature. Zeners can be made from one to several hundred volts. They are used as voltage regulators and voltage reference standards.

Circuit Diagram

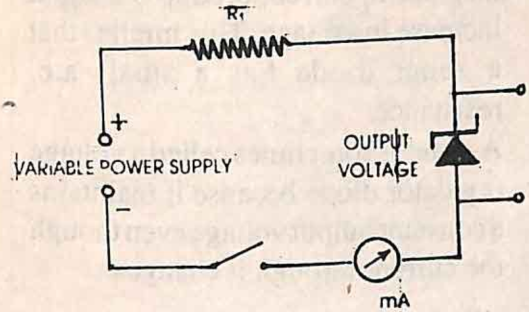


Fig. 5.3

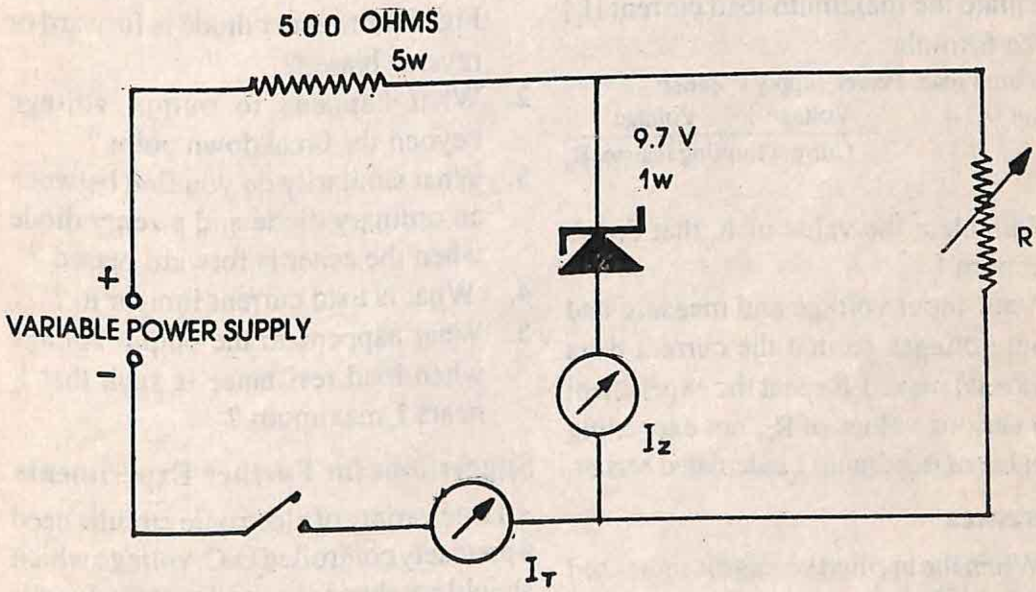


Fig. 5.4

Materials Required

1. Zener Diode- 9.7 V-one
2. Resistor 500 ohms - 6 watt-one
3. D.C. Power Supply (variable) -one
4. Connecting wire etc.
5. Switch—one.
6. Variable Resistor-one

Procedure (Part I)

1. Make the connections as shown in Fig. 5.3.
2. Increase the variable power supply (input volts) in 1 V steps, upto 10 volts- maximum and measure the output voltage in each step. (Note : Never connect a Zener Diode directly to a power supply. Make sure that a current limiting register such as R_1 is included in the circuit).
3. Reverse the connection of the power supply and repeat the experiment.

Observations

When the Zener Diode is reverse biased the current remains zero when the output voltage changes from 0-10V. When the Zener Diode is forward biased the current increases as the input voltage increases.

Procedure (Part II)

1. Make the connections as shown in Fig. 5.4.
2. Increase the power supply to 10 volts.

Observation

Calculate the maximum load current (I_L) by the formula

$$\text{Maximum load current } (I_z) = \frac{\text{Power supply Voltage} - \text{Zener Voltage}}{\text{Current limiting resistor } R_1}$$

Calculate the value of R_L that draws maximum I_L

Vary input voltage and measure and output voltages so that the current does not exceed max. I_L Repeat the experiment with various values of R_L, not exceeding the value of maximum I_L calculated earlier.

Inferences

1. When the applied voltage is increased the diode starts conducting and after a certain value of this voltage, the current increases rapidly. The diode normally does not conduct when reverse biased but if the reverse bias is increased further, then after a certain particular value the diode conducts heavily. This particular value of voltage is called breakdown voltage. However, in the breakdown region an increase in current produces a slight increase in voltage. This implies that a zener diode has a small a.c. resistance.
2. A zener is sometimes called a voltage regulator diode because it maintains a constant output voltage even though the current through it changes.

Questions

1. When the circuit is connected as in Fig. 5.4,the zener diode is forward or reverse biased?
2. What happens to output voltage beyond the breakdown point ?
3. What similarity do you find between an ordinary diode and a zener diode when the zener is forward biased ?
4. What is load current limited to ?
5. What happens to the output voltage when load resistance is such that I_L nears I_z maximum ?

Suggestions for Further Experiments

A large variety of electronic circuits need a precisely controlled D.C. voltage which should not change beyond specified limits during large variation in load. For low power applications a simple shunt regulator using a zener diode of 5 volts is used. The series resistance is choosen to allow a minimum zener current i.e. the current under full load conditions. You may assemble the circuit as given in the Fig. 5.5.

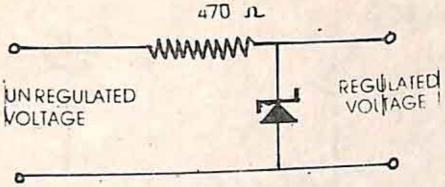


Fig. 5.5

The zener diode circuit may be connected to the output of the full wave rectifier (Expt. No. 4) to get a regulated 5 volts output.

6. Bipolar Junction Transistor

Objectives

To show that a Bipolar Junction Transistor or commonly known as Transistor consists of two p-n junction diodes one between the emitter and the base and the other between the base and the collector.

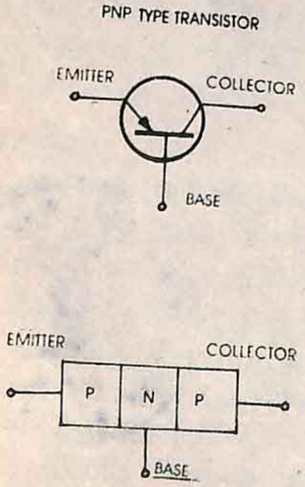
Related Information

Bipolar Junction Transistor is a three terminal device. It is made of semiconducting material containing three separate regions. The middle region is called the base and the two other regions are called the emitter and the collector.

There are two types of transistors.

1. NPN - type
2. PNP - type

The circuit symbols of the two types are shown in Fig. 6.1 and Fig. 6.2.



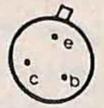


Fig. 6.1

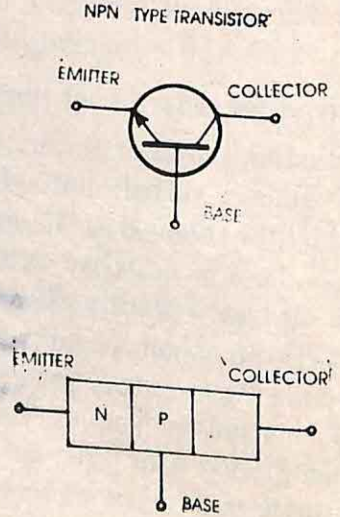


Fig. 6.2

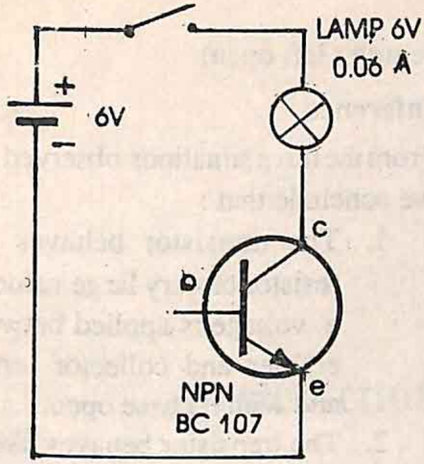


Fig. 6.3

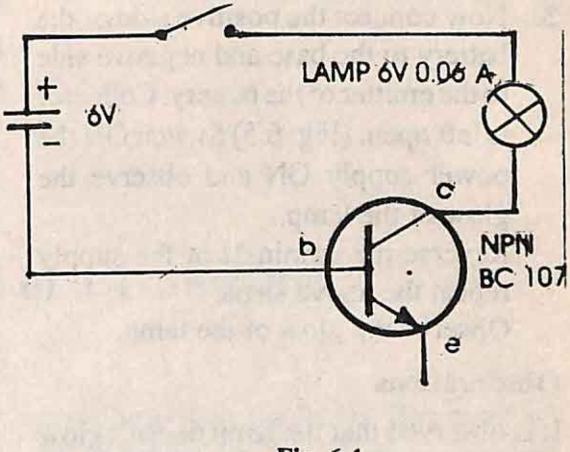


Fig. 6.4

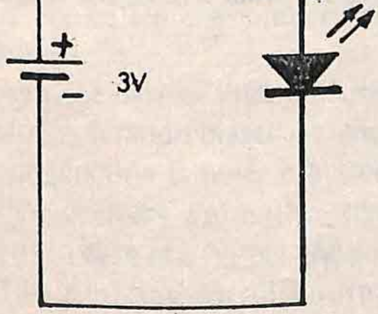


Fig. 6.5

Materials Required

1. Transistor NPN (BC 107) - 1 no.
2. 6 V lamp - 1 no.
3. D. C. supply - 0 - 6V

Procedure

1. Make the connections as shown in Fig. 6.3.

   Collector is connected to the positive terminal of the battery.

   The base is left open and the emitter is connected to the negative side of the battery.

   Switch the power supply ON and observe the glow of the lamp.

2. Now take a 6V battery and connect the negative side to the base of the transistor and positive terminal to the collector of the transistor as shown in Fig. 6.4. The emitter is left open

Switch the power supply ON and observe the glow of the lamp.

3. Now connect the positive side of the battery to the base and negative side to the emitter of the battery. Collector is left open. (Fig. 6.5) Switch ON the power supply ON and observe the glow of the lamp.

   Reverse the terminals of the supply repeat the above steps.

   Observe the glow of the lamp.

Observations

It is observed that the lamp doesn't glow when the collector is connected to the positive and the emitter to the negative side of the battery (the base left open). The lamp glows when the base is positive and the emitter is negative (collector left open). The lamp glows when the base is positive and collector is negative (emitter left open). Again the lamp doesn't glow when the terminals of the supply are reversed. But glows when the base is positive and collector is negative (emitter left open).

Inference

From the three situations observed above, we conclude that :

1. The transistor behaves like a resistor of very large value when a voltage is applied between its emitter and collector terminals and with its base open.
2. The transistor behaves like a p-n junction diode between the base and collector terminals with the base and the p-side and the collector as the n-side in case of a N-P-N transistor.
3. The transistor behaves like a p-n junction diode with the base as the p-side and the emitter as the n-side for the N-P-N transistor.

Questions

If this experiment is repeated with a P-N-P transistor, what will be the findings ?

7. Basic Action of a Transistor

Objective

To show that collector current flow in a transistor only when the base (emitter diode junction is *forward biased*).

Related Information

A transistor is a three terminal device. It is basically made using silicon or germanium semiconductor with either n-p-n or p-n-p configuration. Of the three regions of a transistor the middle region is called the base and the two outer regions are called the emitter and the collector.

When the base-emitter junction of a transistor is forward biased and the base collector junction is reverse-biased, the majority carriers from the emitter-electrons in case of a NPN (and roles in case of a p-n-p transistor) diffuse in to the base region. Since the thickness of the base region is small and also its doping level is small, only a very small fraction of electrons diffuse into the base, recombine with the holes. So, most of the electrons drift to collector region under the action of the applied electric field.

The choice of a NPN transistor has been found to be more suitable because the major part of the current is transported by electrons whose mobility is higher, compared to those of holes.

A transistor is supplied dc-voltages to base collector and base-emitter junctions. The amount and polarities of these voltage depend on the application for which transistor is to be used. The supply of dc-voltages is known as *biasing a transistor*.

In a N-P-N transistor, the base emitter junction is forward biased, when the base is made positive with respect to the emitter (opposite in PNP-type) by connecting the positive terminal of the battery to the base and the negative terminal to the emitter.

Circuit Diagram

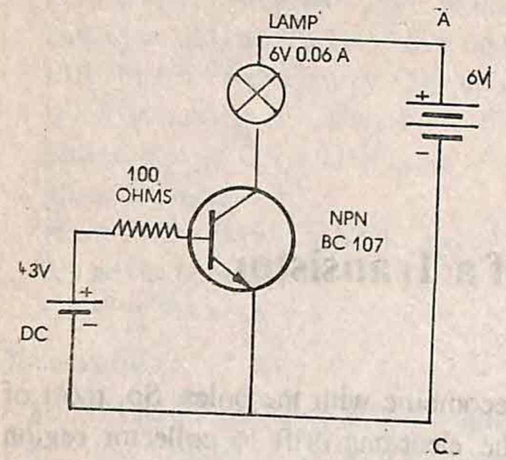


Fig. 7.1

Materials Required

1. Transistor (BC 107) one
2. Resistor of value 100 Ω one
3. LED/ Lamp (6V- 0.06 A) one
4. 6 V power supply or 6V Battery one
5. 3 V Power supply or Battery one
6. SPST Switch one
7. Connecting wires etc.

Procedure

1. Connect the circuit as shown in Fig. 7.1
2. Connect the positive terminal of 6V D.C. supply or battery at A and the negative terminal to C.
3. Connect the positive terminal of the 3 V D.C. power supply or battery at B and the negative terminal to C.
4. Switch the power supply ON and observe the LED/Lamp.
5. Reverse the terminals of the 3 V D.C. supply and observe the LED/Lamp.
6. Switch the power supply OFF.

Observation

It is observed that the lamp glows when the circuit is completed. The lamp does not glow when the terminals of the 3 V power supply are reversed or in other words the base-emitter junction is reverse-biased.

Inference

The N-P-N transistor conducts between the collector and emitter terminals only when its base is slightly positive with respect to the emitter i.e. the base-emitter junction of the transistor is forward biased.

Questions

1. Explain what do you understand by the term "biasing" ?
2. What do you know about the term "forward-biasing" of a NPN transistor ?
3. If the LED/Lamp is removed from the collector circuit and placed in the emitter circuit. What change will you observe, when the power is switched on.
4. Why the resistance of 100 Ω is connected in series with the base ?

8. Transistor as an Amplifier

Objective

To show that a transistors capable of providing current amplification.

Related Information

An amplifier is an electronic circuit that produces an enlarged version of a small signal fed into the circuit. The terminals where the small signal is applied are called the INPUT terminals and the ones where the enlarged signal appears, are called the output terminals.

In a transistor, any change in the base-emitter current produces corresponding change in the collector current, when a load resistance is connected in the collector circuit, the collector current variation will produce a corresponding variation of voltage across the load. This variation of collector current is much more than the variation of base current giving rise to amplification.

Circuit Diagram

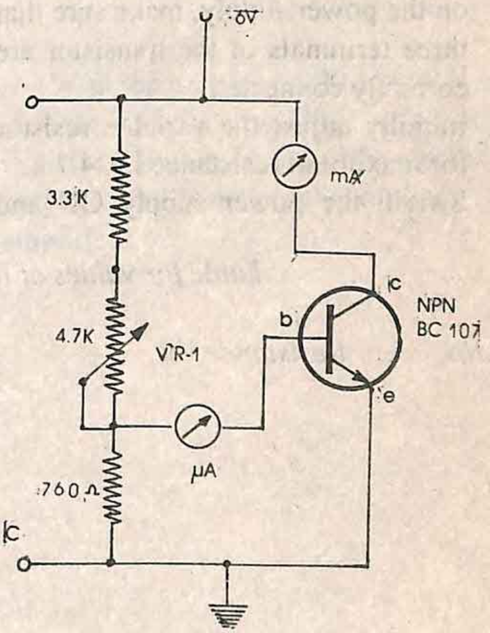


Fig. 8.1

Materials Required

1. Transistor BC 107 one
2. Resistor (760 Ω) $\frac{1}{4}$ w one

3. Resistor (3.3 k)/$\frac{1}{4}$ w one
4. Variable resistor 4.7 K/10 K one
5. 6 V Power supply or 6 V Battery one
6. Micro ammeter, milliammeter and connecting wires etc.

Procedure

1. Connect the circuit as shown in Fig. 8.1.
2. Connect the 6 V Power supply or battery with its positive and to A and negative end to C. Before switching on the power supply, make sure that three terminals of the transistor are correctly connected.
3. Initially adjust the variable resistor for maximum resistance i.e. 4.7 k.
4. Switch the power supply ON and observe the reading on micro ammeter and milliammeter.
5. Now start reducing the resistance of the base circuit by varying the variable resistor.
6. Note down the various readings in ammeter and milliammeter.
7. Switch the power supply OFF.

Observation

Initially, the current does not flow in the circuit, when the circuit is switched ON. When the base emitter voltage is made more than 0.6 - 0.7 V by reducing the resistance, the transistor starts conducting. However a very small current of the order of a few micro amp flows in the base circuit compared to a large current of the order of m A in the collector (output) circuit.

Table for values of input and output current

Sl.No.	*Resistance VR_1*	*Input current (I_b) in μA*	*Collector current (I_c) in mA*

Inference

The current in the collector circuit is very much larger than the current in the base circuit. So transistor is capable of providing current amplification.

Questions

1. How will the circuit of Fig. 8.1 look like when NPN transistor is replaced by a PNP type?
2. Which is more? I_b or I_e (I_b = Base current; I_e = Emitter current). Verify with a suitable current meter.
3. Why a resistance of 760 Ω is in this circuit. Is it necessary? What happens, if this is short-circuited?
4. Why the 3.3 K resister is connected in series with the variable resistor?

Suggestions for Further Experiment

If a CRO is available, then using an audio-oscillator, a small signal a-c amplifier can be made using the circuit given in Fig. 8.2.

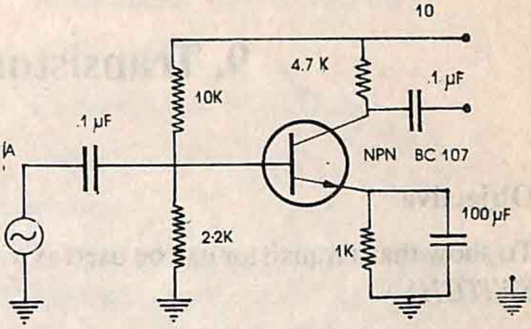


Fig. 8.2

With the help of oscilloscope, it can be observed that the small value input signal at A (input point) and the amplified output signal at the point B (collector of amplifier).

9. Transistor as a Switch

Objective

To show that a transistor can be used as a *SWITCH*.

Relative Information

An ideal switch is one which offers infinite resistance when open and zero resistance when closed. When the transistor is in OFF state, it offers very large resistance conducting only few nano amperes (10^{-9}A) of current and when in ON state, it offers a low resistance of few ohms capable of passing a few milliamp. of current.

Circuit Diagram

In the absence of input voltage, the base current-I_B is zero (Fig. 9.1). No current flows through load resistance R_L except the leakage current I_{CEO} which is very small. In this case, the voltage from collector to emitter is maximum. When a positive voltage is applied between base and emitter, a base current I_B flows, resulting a large collector current. If voltage applied to the base is such that it drives the transistor to saturation, a very large amount of collector current flows. V_{CE} is almost zero. Therefore we get two states (i) when input forward bias voltage is zero the output voltage of collector is maximum (cut-off state) and (ii) when input is given the output is zero (saturation state).

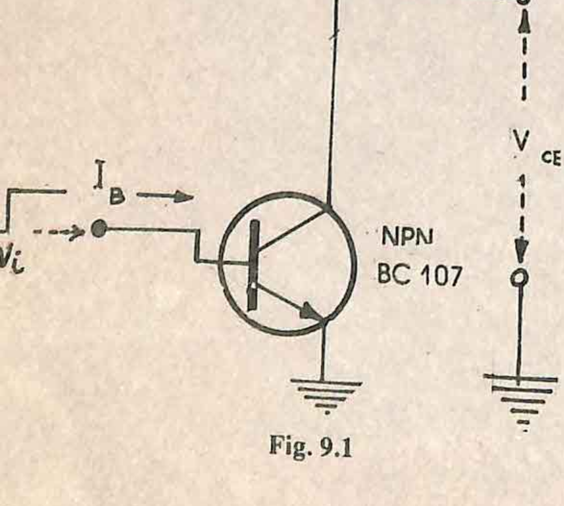


Fig. 9.1

Circuit Diagram

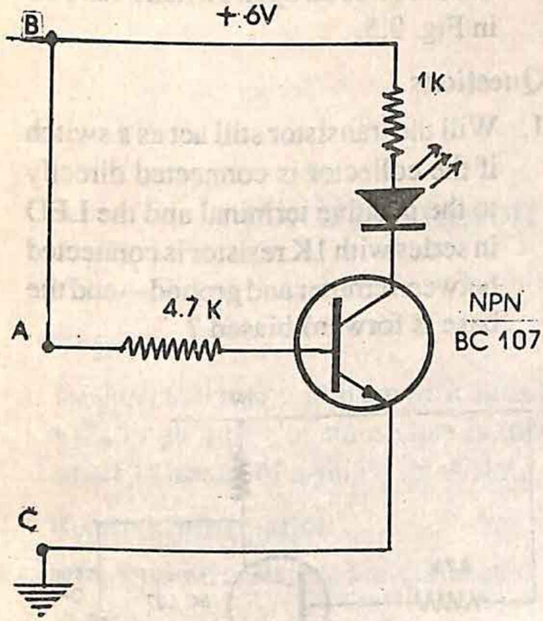


Fig. 9.2

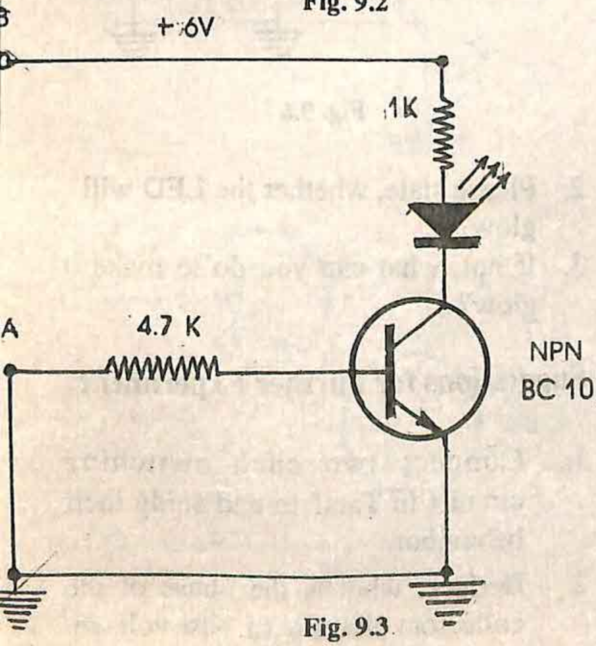


Fig. 9.3

Materials Required

1. Transistor BC-107 — One
2. Resistance 4.7 KΩ /$\frac{1}{4}$ w — One
3. Resistance 1 K /$\frac{1}{4}$ w — One
4. LED — One
5. Connecting wire — One
6. Battery—6V — One

Procedure

1. Connect the circuit as in the Fig. 9.2.
2. Connect point "A" to "B", thus forward biasing the transistor.
3. Observe the glow of LED.
4. Now disconnect A from B.
5. Connect through lead point "A" and "C", so that point "A" is earthed and the transistor is cut-off as in Fig. 9.3.
6. Observe the glow of the "LED".

Observations

1. The LED glows very brightly, when the transistor is forward biased and the collector current is very high.
2. As soon as the forward bias is removed and the transistor is kept at cut-off state, no collector current flows.

Inference

1. The transistor works like a switch, when it is used in saturation and cut-off states. The flow of heavy collector current makes the LED glow. Diagramatically this "ON" state is shown in Fig. 9.4.

2. When the transistor is cut-off it behaves like an open switch as shown in Fig. 9.5.

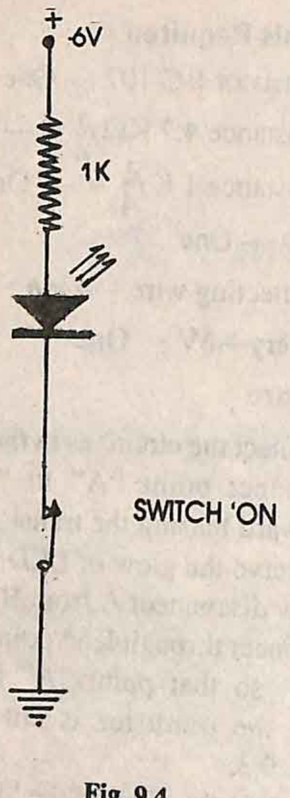


Fig. 9.4

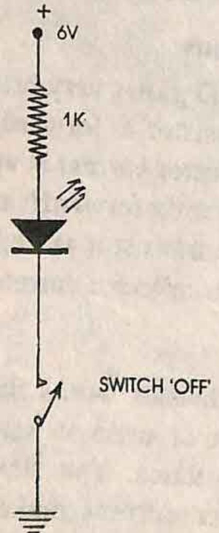


Fig. 9.5

Questions

1. Will the transistor still act as a switch if the collector is connected directly to the positive terminal and the LED in series with 1K resistor is connected between emitter and ground—and the base is forward biased ?

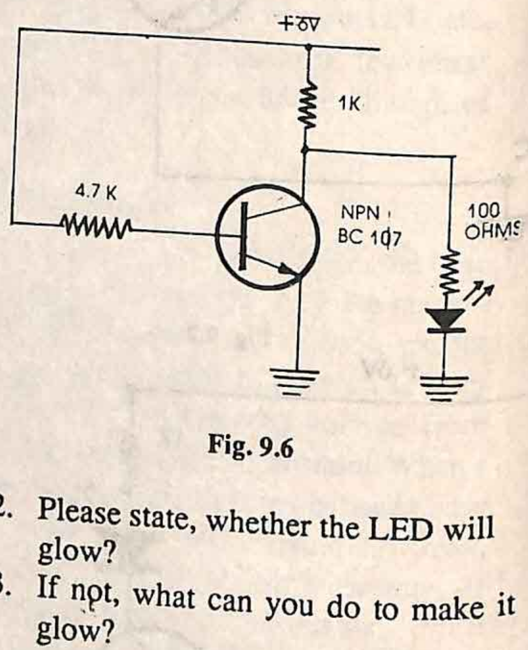


Fig. 9.6

2. Please state, whether the LED will glow?
3. If not, what can you do to make it glow?

Suggestions for Further Experiment

1. Connect two such switching circuits in Tandem and study their behaviour.
2. Deduce, what is the phase of the collector voltage w.r.t. base voltage.

10. Darlington Pair of Transistors

Objective

To show that the current amplification by a Darlington pair of transistors is much larger than that of a single transistor.

Related Information

When two transistors are connected in such a way that the emitter of one is connected to the base of the other, and the collectors of the two are connected together, they are said to form a *Darlington Pair*.

Circuit Diagram

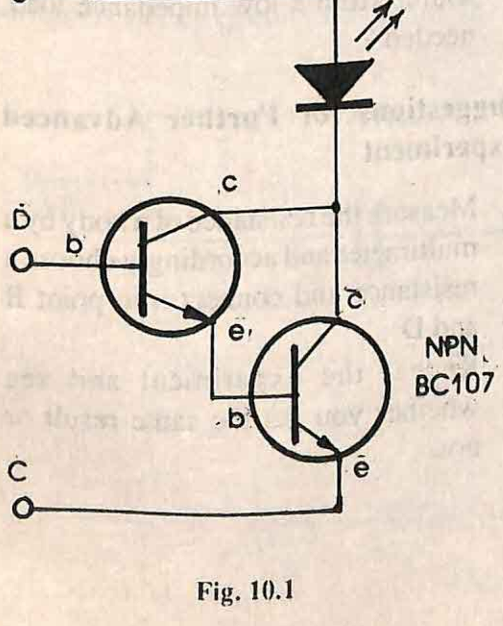


Fig. 10.1

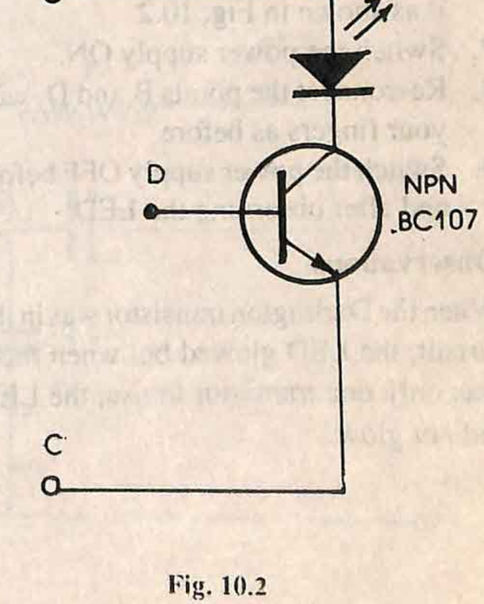


Fig. 10.2

Materials Required

1. Transistor (NPN) BC-107 — Two
2. LED — One
3. 6V DC Power supply or 6V Battery — One
4. Connecting leads with crocodile clips and connecting wires.

Procedure

1. Connect the circuit as shown in Fig. 10.1.
2. Connect the 6V DC power supply or battery with the positive terminal at A and the negative terminal at C.
3. Switch the power supply ON.
4. Connect the points B and D by touching them with your fingers.
5. Observe the LED and switch the supply OFF.
6. Remove the Darlington transistor and connect a simple transistor in place of it as shown in Fig. 10.2.
7. Switch the power supply ON.
8. Re-connect the points B and D, with your fingers as before.
9. Switch the power supply OFF before and after observing the LED.

Observations

When the Darlington transistor was in the circuit, the LED glowed but when there was only one transistor in use, the LED did not glow.

Inference

When the base circuit is completed with fingers, body resistance of a very large value is introduced in the circuit. This results in a very small base current. But the "Darlington transistor" is able to amplify it to a sufficient extent to make the lamp glow brightly. The simple transistor is not able to amplify it to this extent.

Questions

1. What is 'Darlington Pair' of transistor ?
2. What is the input impedance and output impedance of "Darlongton Pair" as compared to a ordinary transistor ?
3. Why the isolation of a high impedance source from a low impedance load, needed ?

Suggestions for Further Advanced Experiment

1. Measure the resistance of a body by a multimeter and accordingly choose a resistance and connect it to point B and D.
2. Repeat the experiment and see whether you get the same result or not.

11. Audio Oscillator

Objective

The objective of this experiment is to construct an "Audio Oscillator", by using a transistor and other circuit components like capacitors and resistors.

Related Information

The sine-wave oscillator uses the principle of positive feed-back, which is a basic requirement of any circuit to oscillate. In a single stage transistor amplifier, the output available at the collector is 180° out of phase with the input at the base. Thus a simple feed-back from the collector to base using a resistance becomes a negative feed-back. The circuit can not therefore oscillate.

A Twin-T filter circuit as shown in the Fig. 11.1 below has been used in the feed-back circuit between collector and base.

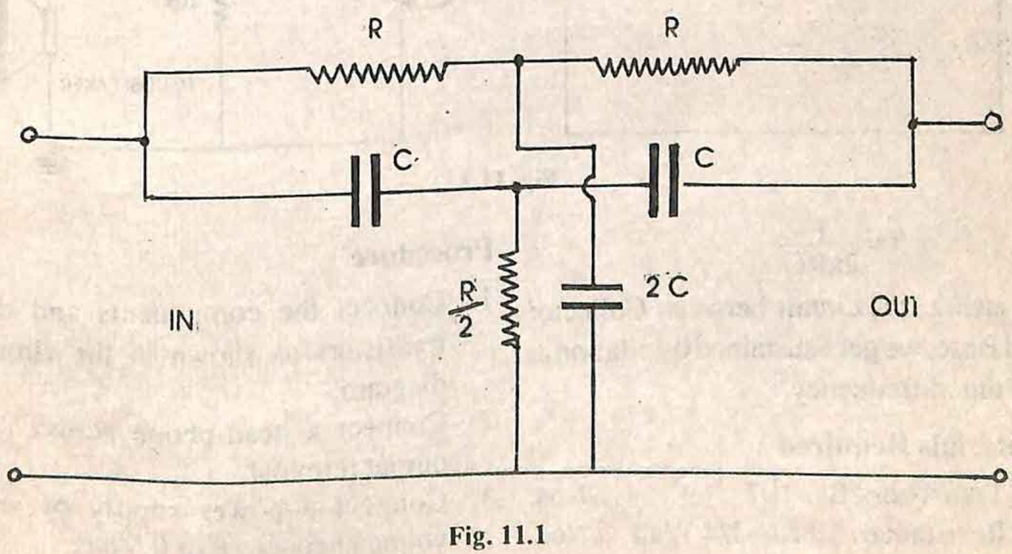


Fig. 11.1

Only at a particular frequency determined by R & C, to which the Twin-T filter is said to be tuned, there is a 180° phase shift between Input and Output. Tune a frequency is given by

$$f = \frac{1}{2\pi RC}$$

By using this circuit between Collector and Base, we get a sustained oscillation at the tuned frequency.

Materials Required

1. Transistors BC-107 2 Nos.
2. Resistances 68 K—1/4 Watt 2 Nos.
3. Resistances 22 K—1/4 Watt 1 No.
4. Resistance 10 K—1/4 Watt 1 No.
5. Variable Resistance 47K 1 No.
6. Condenser 10µF/16V 1 No.
7. Loudspeaker 1 No.
8. Bread-board, Multimeter, Connecting wire etc.

Circuit Diagram

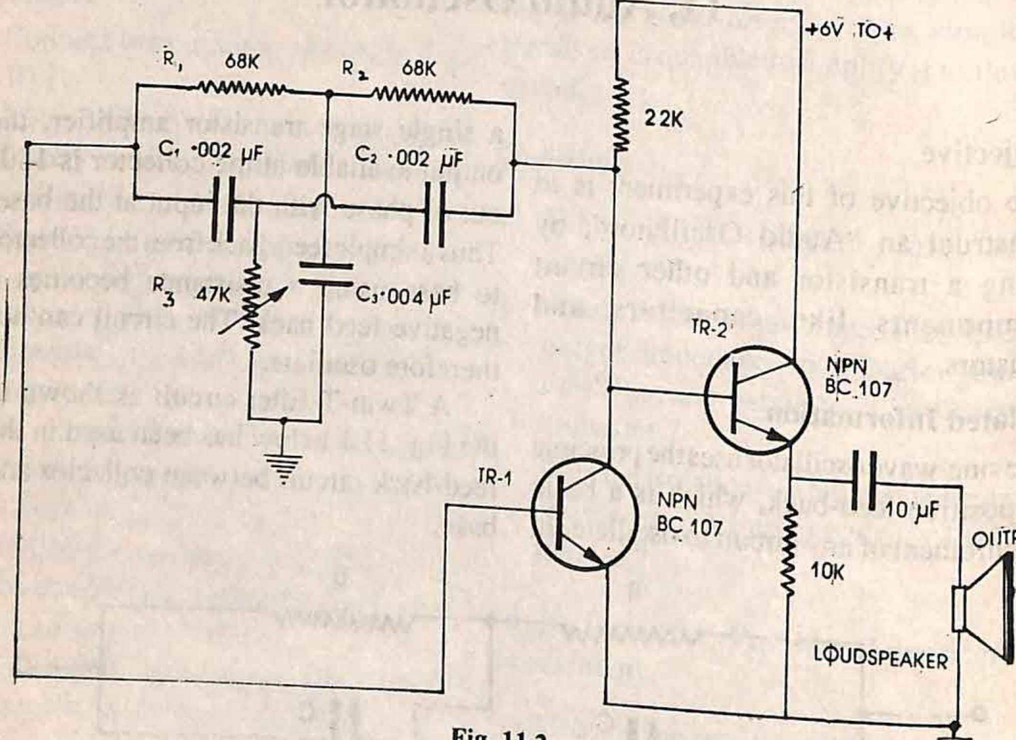


Fig. 11.2

Procedure

1. Connect the components and the transistors as shown in the circuit diagram.
2. Connect a head-phone across the output terminal.
3. Connect a power supply of any voltage between 6 or 9 Volts.

4. Vary the Potentiometer of 47 K gradually.

Observation

1. We hear a tone on the head-phone or Loudspeaker at a particular setting of the potentiometer.
2. Quite-often we have to slightly adjust R-3, to initiate oscillations by what is sometimes called, tuning the Twin-T circuit.

Inference

1. A circuit osciliates, when there is a positive feedback. In other words, when the signal feed-back from collector to the base, is in the same phase as the base voltage.
2. A Twin-"T" filter circuit can be tuned. By varying simultaneously either "C" or "R", the circuit can be made to oscillate over a large range of frequencies.

Questions

1. Under what condition, can a single stage amplifier oscillate?
2. In your circuit, why did you have to use a second transistor.
3. Find out, other ways of positive feed-back, to build-up oscillations.

12. RC Coupled Audio Amplifier

Objective

To study the operation of a two-stage RC coupled audio amplifier.

Related Information

Two or any number of amplifiers operated in cascade may be considered as a single amplifier unit with one input point and a one output point.

When two or more amplifiers are operated in cascade, the characteristics of the total unit must conform to the requirements of the application. For example, if two or more transistor amplifiers in cascade constitute an audio amplifier, the amplifier must be operated over its linear characteristics for distortionless reproduction of sound.

Materials Required

1. Transistors (BC 107) — Two

Circuit Diagram

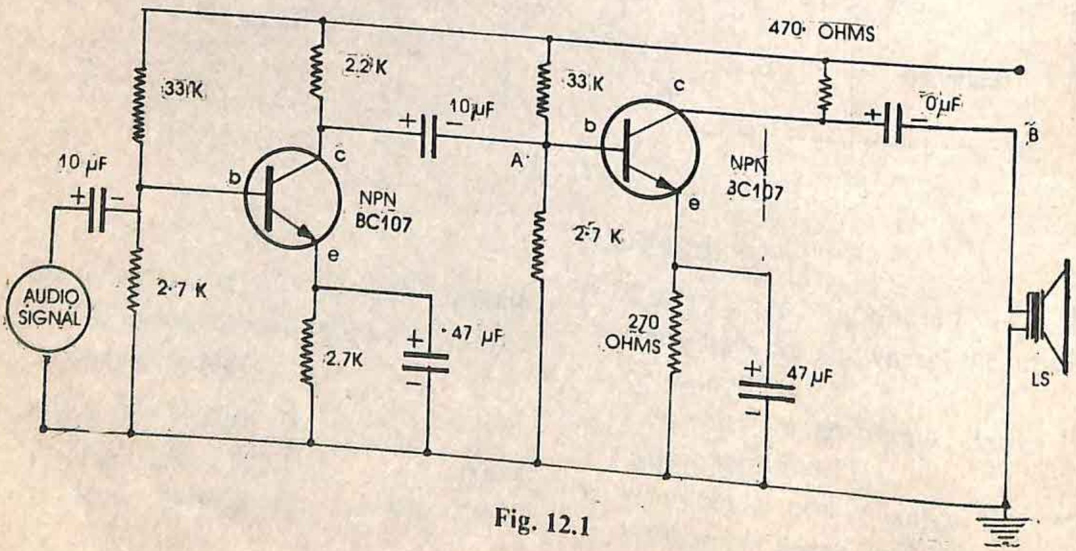


Fig. 12.1

2. Resistance (33 KΩ— Two), (2.2 KΩ – One) (2.7 KΩ – Two), (270Ω – One), (470 Ω – Two)
3. Electrolytic capacitor (10 ufd – three), (47 ufd – two)
4. Power supply (6V) —One
5. A.C. signal generator — One
6 Connecting wires

Procedure

1. Make the circuit as shown in Fig. 12.1.
2. A loudspeaker serves as the load for Q_2. An a.f. signal generator set at 1000 Hz minimum output is used as the signal source.
3. Switch the power supply ON and listen to the loudspeaker.

Observation

If we connect the loudspeaker at point A we can hear a low sound. If we connect the loudspeaker at point B the sound from the loudspeaker is relatively louder.

Inference

The circuit is working as RC coupled, two stage audio amplifier.

Questions

1. What is the purpose of cascading amplifiers?
2. Is the single stage amplifier as effective as the two stage amplifier? If not, then why?

Suggestions for Further Experiments

Connect the output of audio oscillators (done in the previous experiment) to the input of RC coupled, two stage audio amplifier and verify whether you are getting amplified audio signal from the loudspeaker or not.

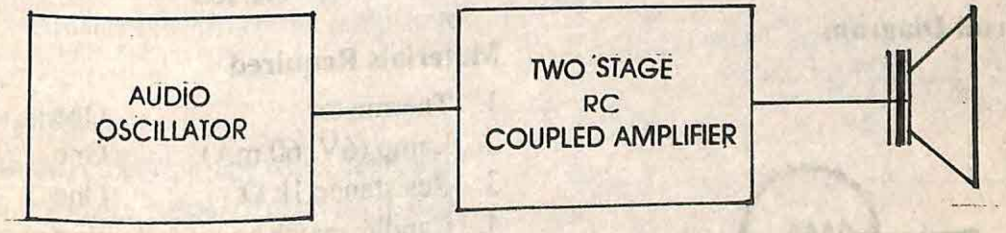


Fig. 12.2 Block Diagram

13. Thermistor

Objective

To show that the resistance of a thermistor decreases with increase in its temperature.

Related Information

Thermistor is made of semi-conducting material and its resistance changes with temperature. The thermistor used in this experiment shows a decrease in its resistance when its temperature is increased. It is said to have a negative temperature coefficient.

Symbol of thermistor   Number of terminals =2

Circuit Diagram

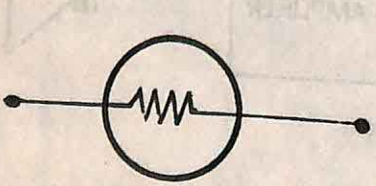


Fig. 13.1

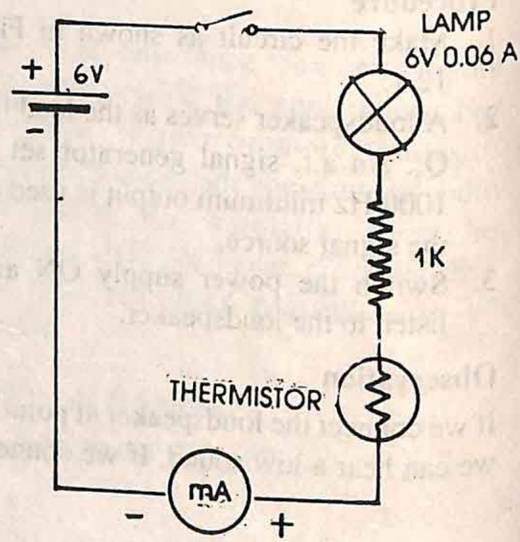


Fig. 13.2

Materials Required

1. Thermistor — One
2. Lamp (6V, 60 mA) — One
3. Resistance 1k Ω — One
4. Candle, match box, connecting wires and multimeter
5. Switch ON/OFF — One

Procedure

1. Make the connections as shown in the Figure 13.2.

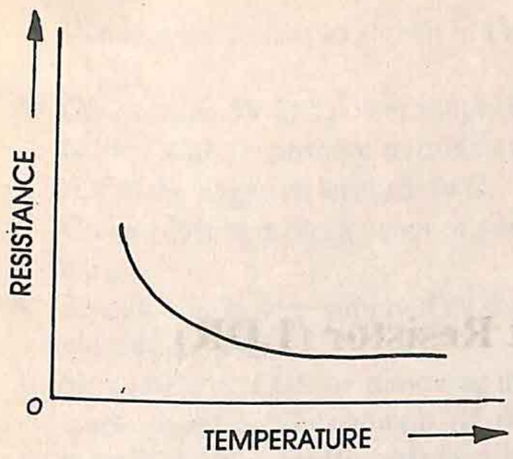


Fig. 13.3

2. Switch the power supply ON.
3. Observe the lamp.
4. Now warm the thermistor with a candle.
5. Watch the lamp again.
6. Switch the power supply OFF.

Observation

The lamp does not glow in the first case, even though the circuit is complete. When the thermistor is warmed the lamp starts glowing.

Inference

The thermistor is a temperature dependent resistor with its resistance decreasing with, increase in temperature i.e. it has a negative temperature co-efficient (NTC).

Questions

1. Define temperature coefficient of a resistance.
2. What would happen to a resistor (with positive temp. co-efficient) when its temperature is increased.

Suggestions for Further Experiments

1. Can you think of making a thermometer using a thermistor?
2. In the above circuit of Fig. 13.2, remove the lamp and close the circuit. Then connect the lamp in parallel with the thermistor and see what happens, when the thermistor is warmed.
3. A thermistor can be used to give warming of temperature changes and operate as an alarm. As shown in Fig. 13.4 the alarm is a LED, but in a fire control system, the alarm can be a bell.

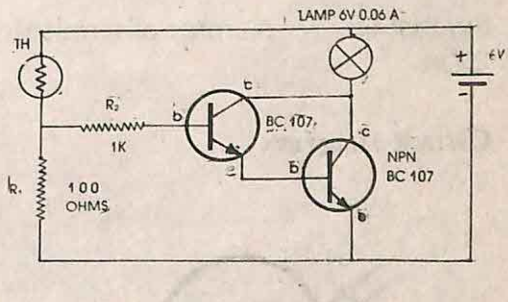


Fig. 13.4

The two transistors here form a "Darlington Pair Amplifier" which you have already studied.

14. Light Dependent Resistor (LDR)

Objective

To show that the resistance of a light dependent resistor (LDR) decreases when a beam of light falls on it.

Related Information

A light dependent resistor is a photo sensitive device, made up of a semi conducting material. The resistance of a LDR decreases when light falls on it.

Symbol of LDR   Number of terminals–2

Circuit Diagram

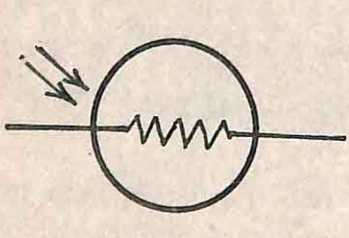


Fig. 14.1

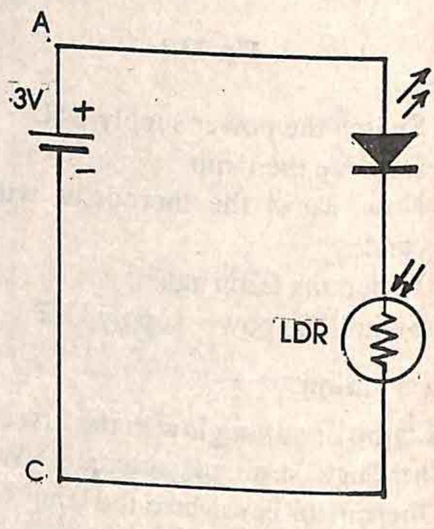


Fig. 14.2

Materials Required

1. LDR — One
2. LED — One
3. 3V DC Power supply or 3Vbattery — One
4. Switch — One
5. Connecting wires etc.

Procedure

1. Connect the circuit as shown in Fig. 14.2.
2. Connect the 3V-DC power supply or battery with its positive terminals at A and the negative terminal at C.
3. Cover LDR by a black paper or your thumb.
4. Switch the power supply ON and observe the LED.
5. Shine light on LDR by removing the black paper or your thumb (If the room light is not sufficient, then by using a pen torch) and observe the LED.
6. Switch the power supply OFF.

Observation

Initially the LED does not glow. When light is thrown on LDR, the LED glows brightly. The resistance of a light dependent resistor (LDR) decreases when light falls on it.

Inference

With the increase in intensity of light falling on LDR, the glow of LED becomes much brighter. This shows that the LDR resistance decreases with increase in intensity of light falling on it.

Questions

1. What is a LDR?
2. If instead of a "LED", a resistance is used, will the LDR behave in the same way?

Suggestions for Further Experiments

1. The LDR can be used in a transistor circuit to switch ON a lamp when light falls on the LDR.

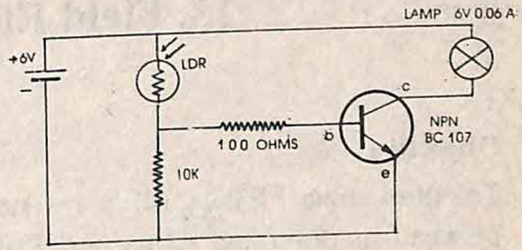


Fig. 14.3

2. You may also assemble the circuit using a LDR and a Darlington Pair as shown below. The lamp in the circuit can be replaced by a relay switch and the device can be used to drive a load.

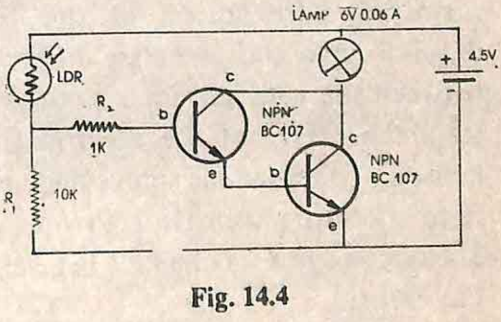


Fig. 14.4

15. Field Effect Transistor

Objective

To show that FET is (a) 'a resistor between its drain and source terminals (b) a p - n junction diode between its gate and either of the other two terminals and (c) the gate source junction of FET has to be reverse biased to control effectively its drain current.

Related Information

Field effect transistor (FET) has three terminals — called the source (S), the brain (d) and the greater (g) current (Id) flows into the FET through the brain terminal and comes out by the source terminal. The action of the FET depends upto the potential difference between the gate and the source (V_{gs}). It is the effect of the electric field between gate and the source that make field effecting transistor to operate. The circuit symbols of a FET is given in Fig 15.1 (i)

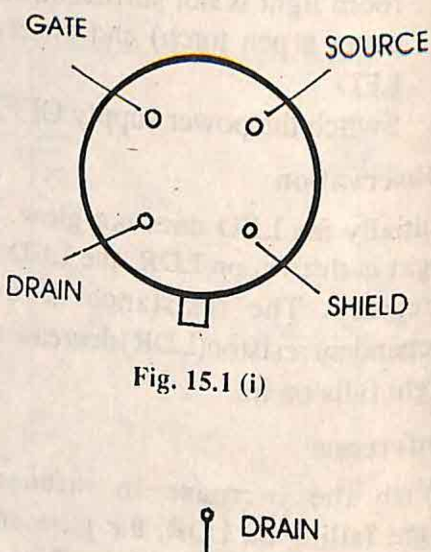


Fig. 15.1 (i)

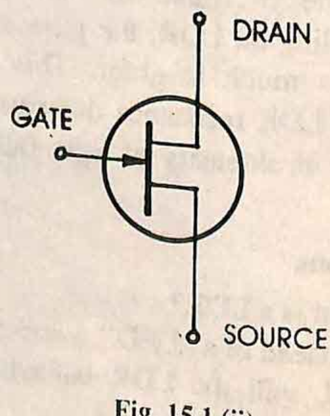


Fig. 15.1 (ii)

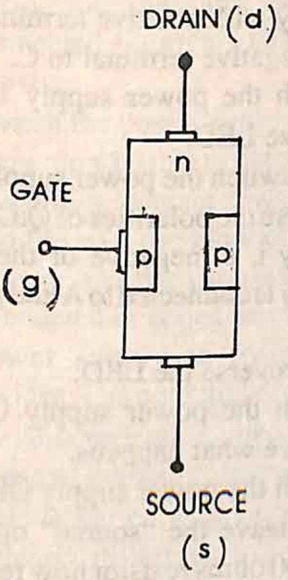


Fig . 15.1 (iii)

Circuit Diagrams

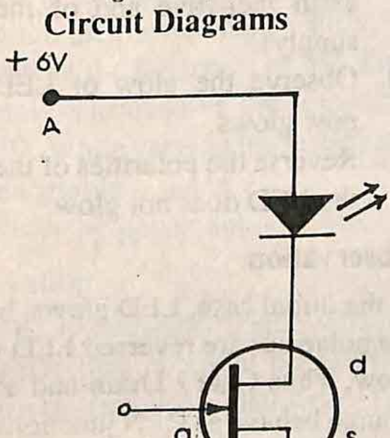


Fig. 15.2

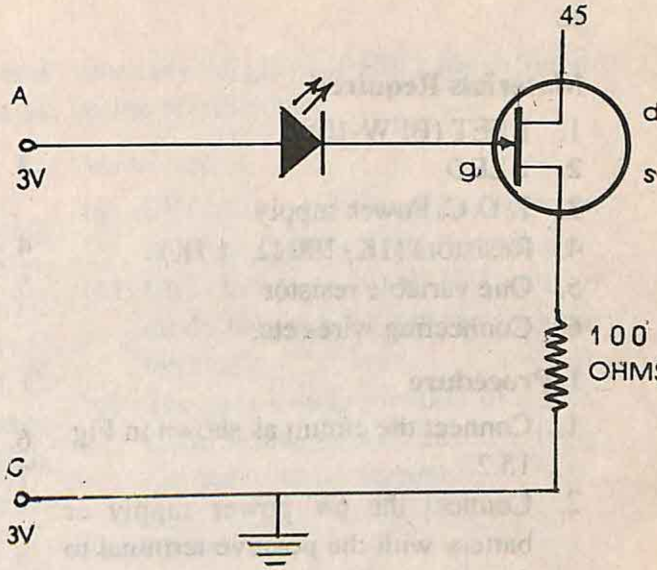


Fig. 15.3

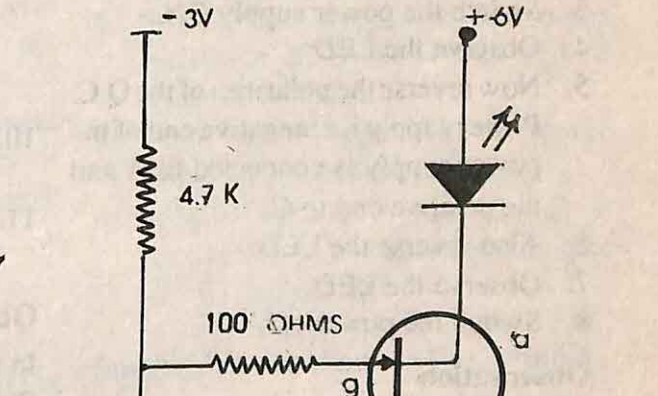


Fig. 15.4

Materials Required

1. 1 FET (BFW-10)
2. 1 LED
3. 1. D.C. Power supply
4. Resistors (1K, 100 Ω, 4.7K)
5. One variable resistor
6. Connecting wires etc.

I. Procedure

1. Connect the circuit as shown in Fig. 15.2.
2. Connect the 6V power supply or battery with the positive terminal to A and negative terminal to C using crocodile clips.
3. Switch the power supply ON.
4. Observe the LED.
5. Now reverse the polarities of the Q.C. Power supply i. e. negative end of the power supply is connected to A and the positive end to C.
6. Also reverse the LED.
7. Observe the LED.
8. Switch the power off.

Observation

The LED glows even when the polarities of the D. C. power supply is changed or "drain" and "source" terminals are interchanged. This is because, the FET behaves as a pure resistance between D & S with Gate "G" open.

II. Procedure

1. Connect the circuit as shown in Fig. 15.3.
2. Connect the 3V power supply or battery with positive terminal to A and negative terminal to C.
3. Switch the power supply ON and observe LED.
4. Now switch the power supply OFF.
5. Reverse the polarities of Q.C. power supply i. e. negative of the power supply is connected to A and positive to C.
6. Also reverse the LED.
7. Switch the power supply ON and observe what happens.
8. Switch the power supply OFF.
9. Now leave the "source" open, put the 100 ohms resistor now removed, in series with Drain and connect to earth (negative and of the 3 volt supply).
10. Observe the glow of LED which now glows.
11. Reverse the polarities of the battery the LED does not glow.

Observation

In the initial case, LED glows, but when the polarities are reversed LED does not glow. Thus Gate - Drain and also Gate Source behave as P - N junctions diodes.

III. Procedure

1. Connect the circuit as shown in Fig 15.4.
2. Connect a 3 V DC power supply such that the negative is connected to B and the positive to C or earth point.
3. Connect another 6V power supply as shown, with its negative end at

earth potential and positive end connected through LED to drain as shown.

4. Switch the power supply ON.
5. Vary the variable resistor by moving the sliding contact towards the high and observe the glow of LED. 4.7K resistor and the variable resistor are connected in series across 3V. D. C. power supply to form a voltage divider. The potential drop across the variable resistor applied between the gate and the source terminals of FET. This is varied to give values of the potential difference from OV to max. value. This varying voltage is used to reverse bias the gate source junction.
6. Switch the power supply OFF.
7. Reverse the polarities of 3V. Power supply and observe the intensity of LED. While varying the resistance of the variable resistor.
8. Switch the power supply OFF.

Observation

Changing the resistance thereby changing the reverse bias potential between gate and source change the intensity of LED. Increasing the resistance decreases the intensity of LED and vice-versa. After the reversal of polarities of 3V. D. C. power supply there is no variation in the intensity of glow of LED for different value of resistance.

Inference

(a) FET behaves like a resistor between its drain and the source terminals.

(b) FET behaves as P–N junction diode between its gate and source terminals.

(c) The gate source junction of a FET has to be reverse biased to effectively control its drain current.

Questions

1. What is the main difference between a junction transistor and FET?
2. What would happen in the source resistance of 1K Ω in Fig. 15.4 is replaced by 5K Ω.
3. What would happened if the source resistance is short circuited ?
4. Why a resistance of 100 Ω is connected in series with the gate of FET in Fig. 15.4 ?

Suggestion for Further Experiment

In the second part of the experiment prove that FET behaves as a P-N junction diode between its gate and drain terminal by disconnecting the source (s) from the power supply and connecting that end to drain (d) through a 100 Ω resistor.

16. Field Effect Transistor (FET) as an Amplifier

Objective

To show that FET is capable of providing amplification and calculate the amplification factor (μ).

Related Information

It is the effect of the electric field between gate the source that makes the field effect transistor to operate. Any change in the gate voltage VGs causes change in the drain current I_D. In a FET amplifier, these changes are converted into large voltage changes by a load resistor R_L.

Circuit Diagram

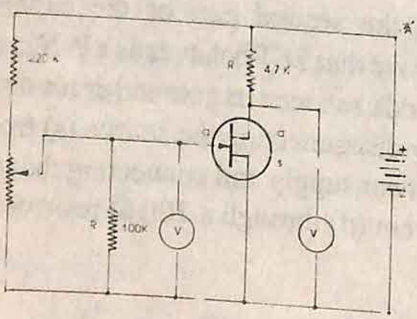


Fig. 16.1

Materials Required

1. 1 FET (BFW - 10) — One
2. Resistors (220 K, 100K4.7K. one each) — One
3. variable resistor (0-4 7K) — One
4. 6V. D.C. power supply — One
5. connecting wires — One

Procedure

1. Connect the circuit as shown in Fig. 16.1.
2. Connect the 6V. power supply or battery with its positive end to A and negative end to C (before switching ON the power make sure that the three terminals of FET are correctly connected).
3. Initially make the resistance of the variable resistor to be maximum i.e. 47K Ω
4. Switch the power supply ON.
5. Now change the resistance of the variable resistor say 2KΩ and note the corresponding voltages V_1 and V_2.

6. Go on changing the resistance and note the corresponding voltages V_1 and V_2.

Observation

The voltage V_1 is less than the voltage V_2. Table for various values of input and output voltages

S. No.	*Variable resistor (in KΩ)*	*Gate Voltage (V_1) in volts*	*Output voltage (V_2) in volts*

Amplification (μ) Factor $\dfrac{V_2}{V_1}$

Inference

A small change in gate voltage produces a considerably large change in output circuit voltage showing that FET is capable of providing amplification.

Questions

1. What is the advantage of a FET amplifier over BJT amplifier?
2. What would happen if you replace the 100KΩ resistor by a 10KΩ resistor?
3. Why 220KΩ resistor is connected in series to the variable resistor?
4. If you connect a 10KΩ resistor in the source circuit, how will it affect the output voltage and consequently the amplification?

Suggestions for Further Experiment

If you have an audio oscillator and a CRO, then you can make a small signal (a.f.) amplifier using FET by connecting the circuit as shown below in Fig. 16.2.

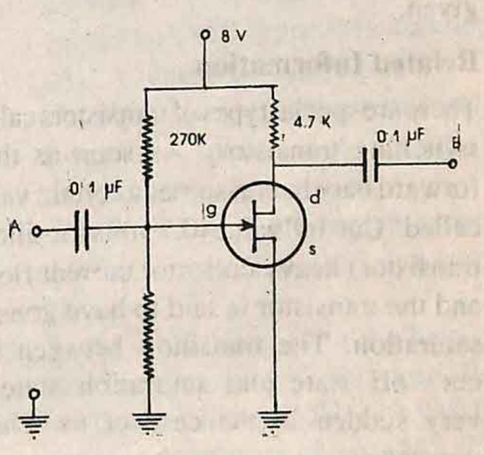


Fig. 16.2

With the help of CRO, you can observe the input signal voltage at point A (input point) and then observe the amplified output signal at point B (output point).

17. Monostable Multivibrator

Objective

The aim of this experiment is to establish that a Monostable Multivibrator has only stable state. This stable state can be disturbed, when an external trigger is given.

Related Information

There are special types of transistors called switching transistors. As soon as their forward base bias assumes a certain value called "Cut-In" value (0.7 volts for silicon transistor) heavy collector current flows and the transistor is said to have gone to saturation. The transition between the cut off state and saturation state is very sudden in the case of switching transistors.

Two transistors, can be inter connected in a number of ways, to result in square wave outputs, which can be repetitive, similar to a vibratory function. Such circuits are therefore called "Multivibrators". The circuit configuration can be several and the one in which there is only one stable state is called "Monostable Multivibrator". When disturbed, it goes through an unstable state, but soon comes back to its stable state.

Circuit Daigram

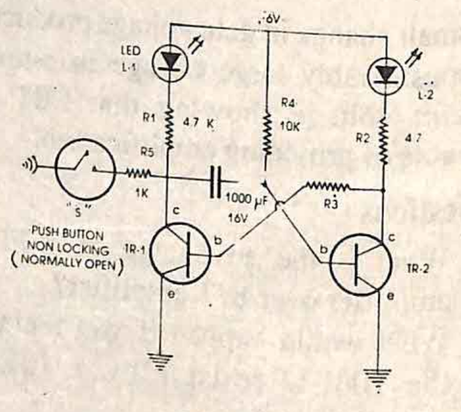


Fig. 17.1

Materials Required

1. Transistor type BC-107 — 2 Nos.
2. Resistors 4.7 KΩ - 1/4 watt — 3 Nos.
3. Resistor 10 KΩ- 1/4 watt — 1 No.
4. Electrolytic Condenser 1000 µF/16V — 1 No.

5. Non locking type push button switch 1 No.
6. 6 volts Power Supply Unit 1 No.
7. Bread- board, Multimeter, etc.

Procedure

1. Connect the different circuit elements as shown in the circuit diagram. (Fig. 17.1)
2. Switch on the +6 volts supply. Observe that the LED (L-2) in the collector circuit glows, and (L-1) does not glow.
3. Activate the push button switch "S" momentarily.
4. Led (L-1) glows and (L-2) goes off, but after sometime again (L-1) goes off and (L-2) glows as before.

Observations

1. We observe that in the stable state, TR-2 remains "ON" and L-2 glows.
2. When a trigger comes, by closing "S" momentarily, the collector of TR-1 abruptly goes to zero potential.
3. This abrupt change in potential at one end of condensor "C" is immediately coupled or passed on to the other side and to the base of TR-2.
4. TR-2 being NPN, it goes to cut off and L-2 goes off.
5. When TR-2 goes to cut off, its collector voltage suddenly goes positive and this positive voltage is directly coupled to the base of TR-1.
6. TR-1 thus stays in a highly conducting state called "saturation".
7. This unstable state of the circuit configuration continues.
8. The condensor "C" in the meantime starts charging, from the 6V, through "R" and through the collector emitter resistance of TR-1, which is virtually zero, because of saturation.
9. As soon as the junction point of C&R which is also connected to base of TR-2, reaches a sufficient positive value—called the cut-in value (0.6V) TR-1 goes to saturation and L-2 starts glowing again.
10. The circuit has returned to its stable state, after having gone through a transitional unstable state.
11. The duration of the unstable state depends upon the time constant "RC".
12. In the stable state, with the help of the multimeter, if we note the collector voltage of TR-2, it will be zero. As soon as we operate "S", this voltage goes to +6V and stays at that value for sometimes (unstable period). The voltage again comes back to zero as in the stable state.
13. The above phenomena can also be represented in a timing diagram as given in Fig. 17.2.

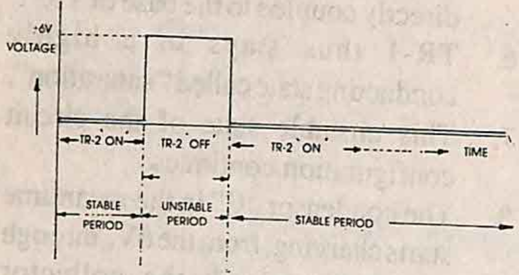


Fig. 17.2 Timing Diagram

In this timing diagram, at time "To", the trigger is applied. The circuit switches over from its stable state to unstable state and stays like that till "T_1", after "T_1" it reverts back to its original stable state. It will continue like that till another trigger is given.

Inferences

From the above experiment, we infer that a monostable multivibrator has only one stable state by an external trigger, it can be made to change over to an unstable state for a short duration. This duration depends upon the time constant R-C. It reverts back to its original stable state, after this short unstable state.

Monostable multivibrators can thus be used as delay circuit to delay certain triggering voltage by a predetermined time. They find wide application in T.V. receiver circuits and other such special equipment involving controlled delay. An example in this regard can be, switching "ON" of the stair case light. Which goes "OFF" automatically after certain present time.

Questions

1. Why is the circuit called a monostable circuit ?
2. Discuss the effects that you will observe by changing the value of "R".
3. Why is it necessary to use switching transistor.
4. Can we use a relay in place of "R_2" as shown in the circuit.
5. Why is it necessary to have a separate –6V supply, as shown in the circuit?

Suggestion for Further Experiment

1. With the help of an Oscilloscope, it would be more interesting to observe the output wave-form at the collector of TR-2, after connecting repetitive trigger pulses through a coupling condenser in place "S". Effect of variation of "R" or "C" can be more effectively observed.
2. With the help of the multimeter, measure the voltages at the collectors of TR-1 & TR-2. Activate "S" switch and measure them again. Note the difference. Also observe, how long it stays like that.
3. Draw a timing diagram in a graphical form by observing the voltage at the collector of TR-1.

18. Bistable Multivibrators

Objective

We will infer from this experiment that the Bistable Multivibrator has two stable states. When any external trigger is applied, the circuit changes from one stable state to another stable state.

Related Information

While conducting the earlier experiment on "Monostable Multivibrator" we have already briefly discussed about switching transistors. We have also seen how two switching transistors can be suitably inter-connected using passive circuit elements like resistors, capacitors to give us a circuit called Multivibrators. A circuit configuration, with two transistors and other R and C elements combined suitably can result in a "Bistable Multivibrator". As the name itself suggests, it has two stable states.

Circuit Diagram

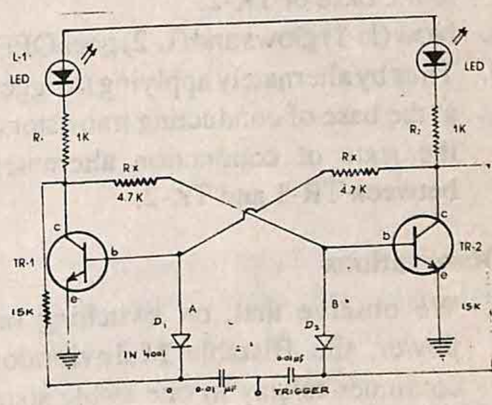


Fig. 18.1

Materials Required

1. Transistors BC 107 — 2 Nos.
2. Resistors 1 KΩ 1/4 watt — 2 Nos.
3. Resistors 4.7 KΩ 1/4 watt — 2 Nos.
4. LEDs — 2 Nos.
5. Bread-board, Multimeter etc.

Procedure

1. Connects the different components according to the circuit diagram.

2. Switch on the Power Supply.
3. With multimeter, measure to ensure that +6V is available from the Power Supply Unit.
4. When switched "ON" either (L-1) or (L-2) will glow. Let us assume (L-1) glows indicating that TR-1 is ON.
5. Momentarily connect to earth the base of TR-1 with the help of the flexible link wire.
6. Now TR-2 conducts, (L-2) glows and (L-1) goes OFF.
7. Next time connect the earth link wire to the base of TR-2.
8. Now (L-1) glows and (L-2) goes OFF.
9. Thus by alternately applying a trigger at the base of conducting transistors, the state of conduction alternates between TR-1 and TR-2.

Observations

1. We observe that, on switching on power, the Bistable Multivibrator continues to stay in one stable state with say TR-1 conducting and (L-1) glowing. TR-2 is off, so also (L-2).
2. Measurement of voltage at the collector of TR-1 gives a very low reading, almost zero.
3. On applying the trigger, the Bistable Multivibrator switches over to its other or 2nd stable state. TR-2 conducts and (L-2) glows. TR-1 goes off and (L-1) does not glow.
4. Again if we apply another trigger, conditions reverse and the circuit goes back to its 1st stable state— where TR-1 conducts and (L-1) glows.

Inference

We thus infer by conducting the experiment and observing the results that the Bistable-Multivibrator has got two stable states.

Only when an external positive going trigger is applied, as shown in the circuit diagram, the multivibrator changes from one stable state to the other. It stays like that till another trigger arrives.

When we connect the multimeter to the collector of TR-1 and assuming that initially it is conducting, we get a reading of "0" volts. When we apply a trigger thus reading goes to '+6' volts, since TR-1 goes to cut off. With the next trigger TR-1 goes to conduction and the reading goes back to '0' volts. Thus the reading changes back to normal on alternate triggers.

We thus, also observe that this circuit can behave as a counter of "2". It is therefore called a "Binary Circuit" and also sometimes called " Flip-Flop Circuit." when a train of repetitive positive going triggers are applied to the bases the observation of wave-forms at the collector of any of the transistors show very interesting results. A timing diagram as shown below, indicates, how each alternate pulse, brings back the original stable state.

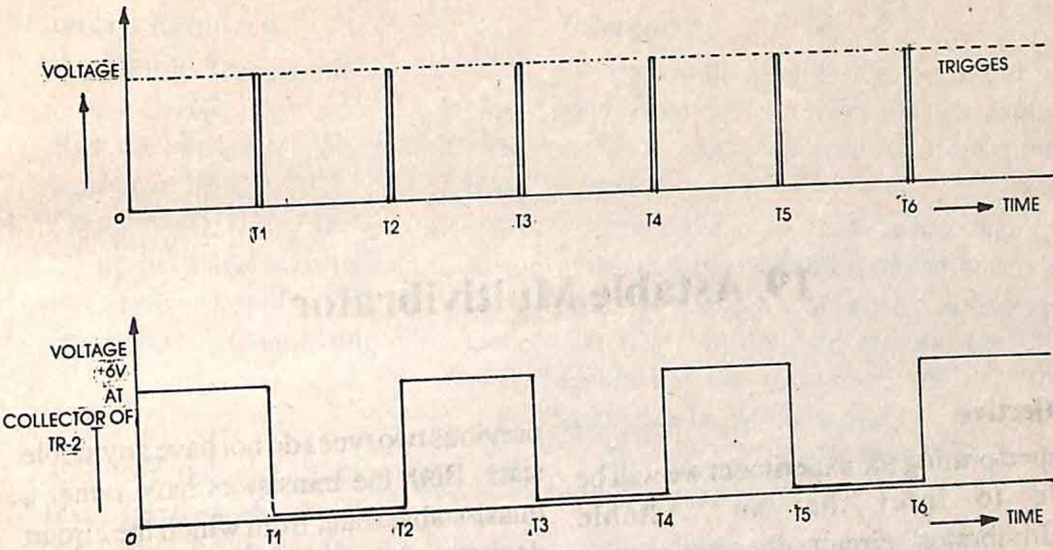


Fig. 18.2

From the above timing diagram, we observe that, at every alternate pulse, collector of TR-2 changes from 0 to +6V. Thus we observe that the transistors change to any one of their stable states—either "ON" or "OFF" state—after every alternate pulse.

It is also called a counter of "2" or Binary and is used in counting circuits, or divider circuits.

Questions

1. As shown in the circuit, suppose we connect the link-wire a negative supply momentarily. What will happen to the two transistors ?
2. Why is the circuit called Bistable ?
3. If we apply only one trigger and nothing else, how long with the circuit take to come back to the initial state ?
4. Why do we use switching transistors in the circuit ?
5. What are the different names for a Bistable Multivibrator ?

Suggestions for Further Experiment

By varying the supply voltage from 6V to say within ± 2V, the performance of the Bistable multivibrator can be studied.

Also by analysing the Oscilloscope wave forms, we might try to alter the circuit elements and study their effects.

19. Astable Multivibrator

Objective

By performing the experiment we will be able to infer that an "Astable Multivibrator" circuit, does not require any external trigger to make the circuit work. Also we will infer that it is like a free running oscillator, and the frequency depends upon the circuit elements.

Related Information

During the previous two experiments on monostable and bistable multivibrator, we have briefly discussed, why such circuits are called multivibrators. Also we have seen that switching transistors are used in multivibrators, specially because they go to saturation state from their cut-off state at a very fast rate. This results in production of square-waves at the output. Square waves are of immense use in pulse circuits, delay circuits, counter circuits and other such special applications.

Astable Multivibrators, unlike the previous two types do not have any stable state. Both the transistors have rather a quasi-stable state, from which the circuit derives it's name as "Astable Multivibrator". Each of the two transistors keep on switching between the "ON" state and "OFF" automatically at a frequency depending upon the RC values. No external triggering is required. They are free-running multivibrators.

Circuit Diagram

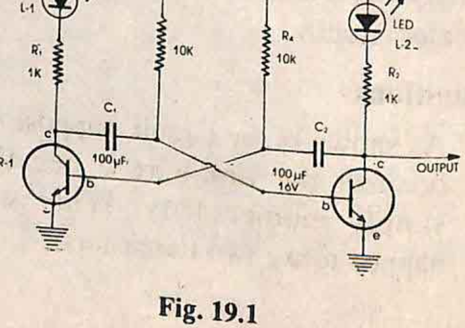


Fig. 19.1

Materials Required

1. Switching Transistors type BC-107 — 2 Nos.
2. Resistors 1 KΩ 1/4 W — 2 Nos.
3. Resistors 10 KΩ 1/4 W — 2 Nos.
4. Condenser 100 μF/ 16 V — 2 Nos.
5. Commonly available LEDs — 2 Nos.
6. D.C. Power Supply 12 V — 1 No.
7. Bread-board, Connecting wires etc.

Procedure

1. Connect the circuit elements and transistors as shown in the circuit diagram. Switch ON the power supply.
2. The LEDs glow ON and OFF at a regular periodicity.

Observations

we observe that, the quasi-stable states of the transistors are exhibited by the LEDs in the collectors going ON and OFF.

Thus none of the transistors stay in a stable state for a very long period.

Also the switching ON and OFF of the transistors is automatic an it behaves as a free-running oscillator.

When we put an oscilloscope (if available) at the collector of any one of the transistor, we observe a train of square-waves.

If we put two resistance of 10 KΩ each in parallel with R_3 and R_4, the LEDs go OFF and ON more quickly, i.e. the frequency of oscillation has increased.

Inferences

By conducting this experiment we have been able to infer that an astable multivibrator is a free-running multivibrator.

We have also concluded that the frequency of oscillation depends on the time-constant R.C., because we observed that if R_3, and R_4 decreases, thereby decreasing the time constant RC, the frequency becomes higher.

The frequency of oscillation of our circuit is given by

$$F = \frac{1}{T} \text{ Where } T = T_1 + T_2$$

and $T_1 = 0.69\,(R_3 \times C_1)$

$T_2 = 0.69\,(R_4 \times C_2)$

When we make $R_3 = R_4$

and $C_1 = C_2$

$T = 1.38\,(R_3\,C_1)$ or $1.38\,(R_4\,C_2)$

It is normally written as $T = 1.38$ RC.

The circuit that we have constructed has time period of roughly 6 secs.

Questions

1. Why is the circuit called Astable ?
2. In place of $R_3 = 10$KΩ, we put 5 KΩ fixed resistance in series with a 5KΩ variable resistance. What effect do you observe, when you vary this resistance ?
3. What would be the required power supply when we use PNP transistors?
4. Can we use one NPN and one PNP transistor ?

Digital Experiments

Introduction of Experiments on Logic Gates

A gate is a digital circuit that follows certain *logical* relationship between the input and output voltages. Therefore, they are generally known as *logic gates*. There are three basic logic gates called OR, AND and NOT. Each basic gate is indicated by a symbol , and its function is defined either by a truth table that shows all possible input combinations and the corresponding outputs or by a Boolean expression. These logic gates can be realised using semi-conductor devices; diodes and transistors.

In computation, it is more convenient to use binany numbers. A binany number has only two digits 0 and 1. We can have a voltage corresponding to 0 (Say 0V) and 1 (Say 5V). In digital electronics we use only two levels of voltages, i.e. 0 and 5V.

Signals are called digital signals. In digital circuits only two values represented by (0 or 1) of the input and output voltage are permissible.

When we use semi-conductor devices for making logic gates, we find that at the output we get a voltage slightly less than 5V or slightly greater than 0, 1 whichever the case, when a 5V dc supply is used. This is because of the fact that there is some voltage drop across the junctions (no diode is practically ideal). Therefore, quite often logic, state 1 is referred to as 'high' and logic state 0 as 'low'. In our practical circuits the negative terminal of the dc source (corresponding to 0V) is taken to be 0 and the positive terminal is taken to be 1.

20. Logic OR-gate

Objective

To study Logic OR-gate using two diodes.

Related Information

The OR-gate has two or more inputs but only one output. It gives a high output if any of the output signal is high.

An OR-gate can be build by two diodes as shown in the Fig. 20.2. If both inputs are low, the output is low. If either input is high, the diode with the high input conducts and the output is high. BOOLEAN EQUATION for OR-operation is —

$Y = A + B$

Symbol for logic OR-gate is given in Fig. 20.1

A

B Y

$Y = A + B$

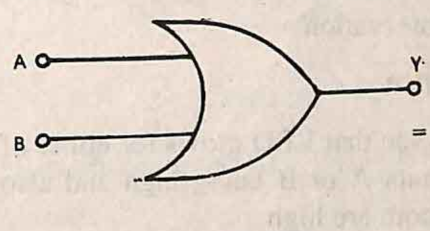


Fig. 20.1

Circuit Diagram

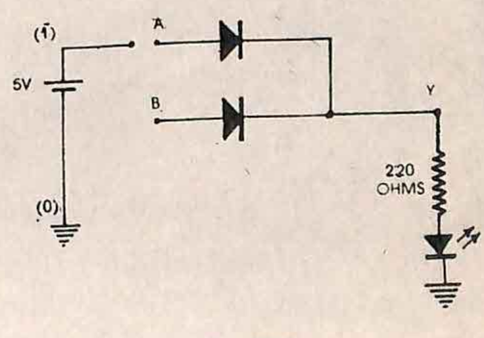


Fig. 20.2

Materials Required

1. Diodes (IN 4001) Two
2. 5V DC power supply One
3. Resistance (220) One
4. LED One
5. Connecting wires
6. Bread Board

Procedure

1. Make the connection in the bread board as shown in Fig. 20.2.
2. Connect A & B to (0), note the state of LED.
3. Then connect A to (1) and B to (0) see whether LED glows or not.
4. Connect A to (0) and B to (1). Record your observation.
5. Connect A and B to (1). Record your observation.

Inference

You'll see that LED glows for either of the inputs A or B being high and also when both are high.

Observation

Input A	*Input B*	*LED**
Low (0)	Low (0)	ON/OFF
High (1)	Low (0)	ON/OFF
Low (0)	High (1)	ON/OFF
High (1)	High (1)	ON/OFF

* Strike off whichever isn't applicable.

Question

1. What will be the output of a 6 terminal OR-gate if at least one of its input is high?

Suggestions for Further Experiment

Construct an OR gate using two switches. Draw the truth table of the above mentioned.

21. Logic AND-gate

Objective

To study Logic AND-gate using two diodes.

Related Information

The AND-gate has two or more inputs but only one output. All inputs must be high to get a high output. When both the inputs are low, both diodes conduct and pull output down to a low voltage. If one of the inputs is low and the other high, the diode with the low input conducts and this pulls the output down to a low voltage. The diode with the high input doesn't conduct. When both the inputs are high, both the diodes do not conduct. The BOOLEAN EQUATION for AND operation is Y = A. B

Symbol of an AND-gate is given in Fig. 21.1.

A

Y = A.B

B

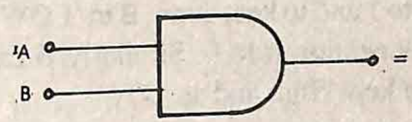


Fig 21.1

Circuit Diagram

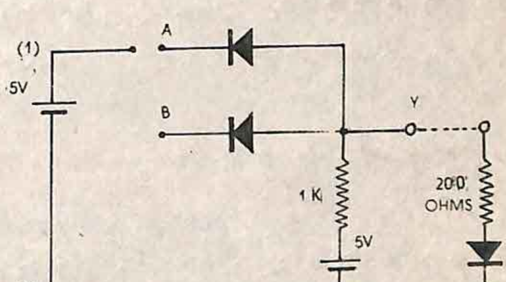


Fig 21.2

Materials Required

1. Two diodes (IN 4001)
2. Resistors (1K Ω , 220 Ω) one each
3. 5V battery One
4. Connecting wires, soldering iron
5. Crocodile clips

Procedure

1. Make the connections as shown in Fig. 21.2.

2. The two inputs A & B can be maintained in either logic state '1' or logic state '0' by connecting A & B to either 1 or 0 respectively.

3. To keep to input A 'HIGH', connect it to 1 and to keep input B to 'LOW', connection it to 0. Similarly B can be kept High and as LOW.

4. The 'HIGH' or 'LOW' outputs will be known by the state of (ON or OFF) LED.

Observations

Input A	*Input B*	*LED**
Low (0)	Low (0)	ON/OFF
Low (0)	High (1)	ON/OFF
High (1)	Low (0)	ON/OFF
High (1)	High (1)	ON/OFF

*Strike of whichever is not applicable.

Inference

The LED glows (i. e. the output is high) only when both the inputs A and B are 'HIGH'.

Questions

1. If an inverter is placed in series with an AND-gate, what will be the result ? (write its truth table only)
2. What is name of that circuit ?
3. Write its symbol also ?

22. Logic NOT-gate

Objective

To show that a circuit for performing the Logic NOT operation can be constructed using a single switch.

Related Information

It is a circuit which has got only one input and one output. It produces a high output (+5V) if the input is low (0V) and vice-versa. That is whatever the input, the output is always inverted. That is why NOT-gate is also known as an inverter.

The Boolean Equation for NOT-gate is $A(Y) = \overline{A}$

Bar over A stands for the complement of A.

Symbol for NOT-gate is given in Fig 22.1 $A(Y) = \overline{A}$

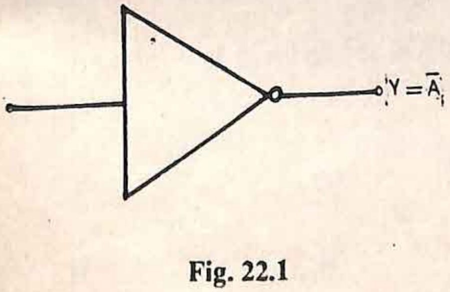


Fig. 22.1

Circuit Diagram

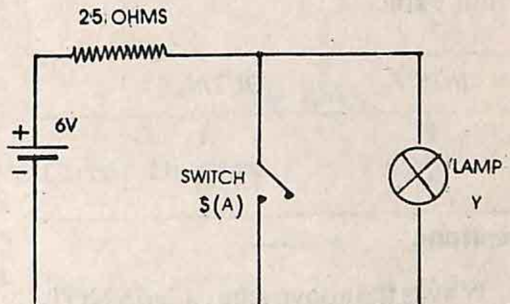


Fig. 22.2

Materials Required

1.	Resistance (25Ω)	One
2.	Slide switch	One
3.	Lamp 6V - 60 mA	One
4.	Power supply 6V	One

Procedure

1. Make the connections as shown in Fig. 22.2.
2. Switch the power supply ON.
3. Observe the conditions of the lamp Y, for the states of the switches.

Observation

The lamp Y is ON, when the switch is not ON (OFF) and vice-versa.

Truth Table

A	*Y*
ON	OFF
OFF	ON

Inference

The logic NOT operation is found to be obeyed. It is usually given in the form of a Truth Table.

INPUT	*OUTPUT*
0	1
1	0

Questions

1. Why is the above gate called a NOT-gate ?
2. What will be the output if we connect two NOT-gate in series?

Suggestions for Further Experiment

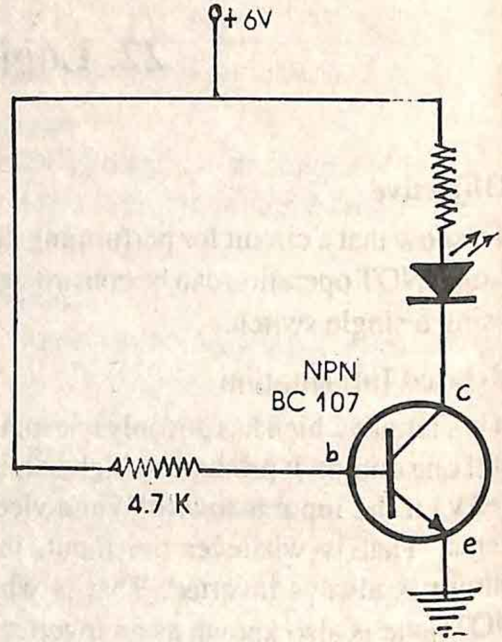


Fig. 22.3

23. Logic NAND-gate

Objective

To study a NAND-gate.

Related Information

The NAND-gate has two or more inputs but only one output. All inputs must be high to get a Low output. A NAND-gate is a combination of AND-gate and a NOT-gate. The output of AND-gate is fed to the input of NOT-gate and the output of NOT-gate is now the output of this combination of gates called NAND-gate.

The Boolean Equation for NAND-gate is

$Y = \overline{A.B.}$ Bar over A. B. suggests that it is the compliment of AND (A.B) operation.

The logical symbol for NAND-gate is

A Y B

A B $Y = \overline{A.B}$

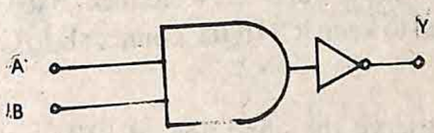


Fig. 23.1

Circuit Diagram

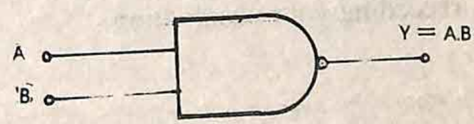


Fig. 23.2

Materials Required

1. Two diodes (In 4001)
2. One transistor (CIL - 147)
3. Resistors (4.7KΩ 50 Ω, 1KΩ)
4. 5V Battery
5. LAMP/LED
6. Connecting wires

Procedure

1. Make the connection as shown in Fig. 23.2.

2. The two inputs A & B can be maintained in either logic state '1' corresponding to 5V (high level) or logic state '0' corresponding to 0V (low level)using lead wires.

3. To keep the input A 'HIGH' connect it to A and to keep this 'LOW' connect A to 0.

4. To keep input B 'Low' connect B to D and to keep it 'HIGH' connect B to C.

5. Observe the condition of LED and make on observation table as shown below.

6. Switch off the power supply after recording your observation.

Observations

Input (A)	*Input (B)*	*Lamp (Y)**
Low (0)	Low (0)	ON/OFF
Low (0)	High (1)	ON/OFF
High (1)	Low (0)	ON/OFF
High (1)	High (1)	ON/OFF

*Strike off whichever is not applicable.

Questions

1. How many inputs can a NAND-gate have?
2. What will be the output of a NAND-gate followed by an inverter?
3. Make the truth table for the above mentioned gate?
4. compare the gate with the AND-gate?

Suggestions for Further Experiment

1. Make a NAND-gate followed by an inverter/two inverters.
2. Another circuit for a two-input NAND-gate is given below:

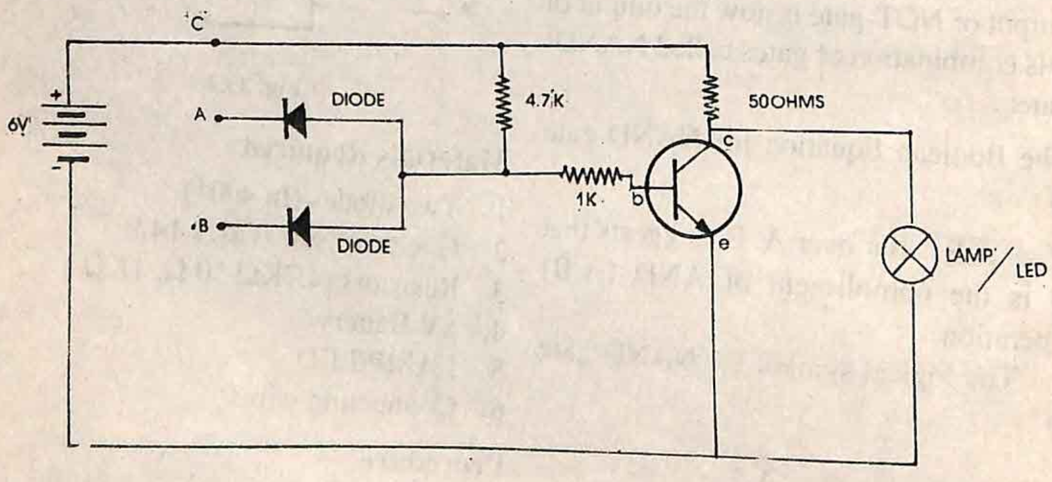


Fig .23.3

Study its operation by alternately connecting input points A & B to High (1) and Low (0) state as done in your experiment.

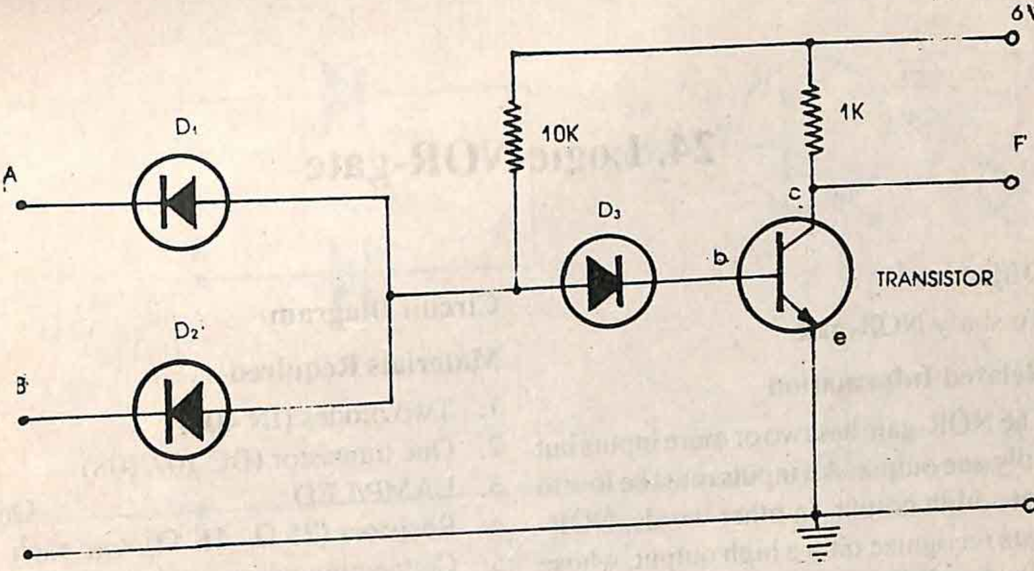


Fig 23.4

24. Logic NOR-gate

Objective

To study NOR-gate

Related Information

The NOR-gate has two or more inputs but only one output. All inputs must be low tó get a high output. In other words, NOR-gate recognize only a high output, whose bits are all O's.

The BOOLEAN EQUATION for a 2-inut NOR-gate is

$$Y = \overline{A+B}$$

Bar over A+B stands for its complement.

Symbol for NOR-gate is

A   A

Y =

B   B $Y = \overline{A+B}$

Circuit Diagram

Materials Required

1. Two diodes (IN 4001)
2. One transistor (BC 107/108)
3. LAMP/LED
4. Resistors (25 Ω, 1K Ω) One one each
5. Connecting wires
6. 5V Power supply (D.C.)

Procedure

1. Make the connection as shown in Fig 24.2.
2. the two inputs A, & B can be maintained in either logic state '1' or logic state '0' using lead wires.
3. To keep the input A high, connect A to C to keep it low, connect A to D.

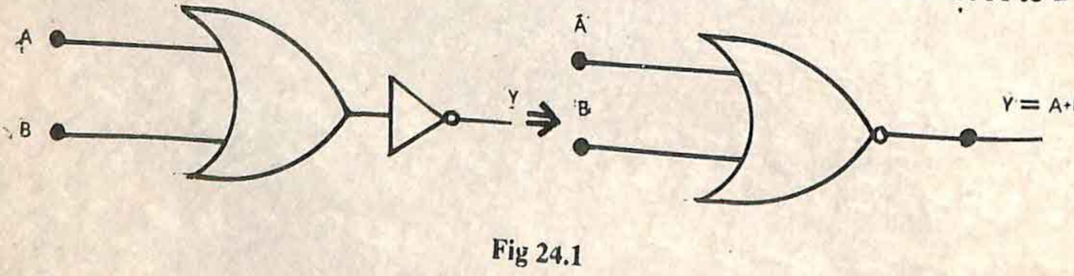


Fig 24.1

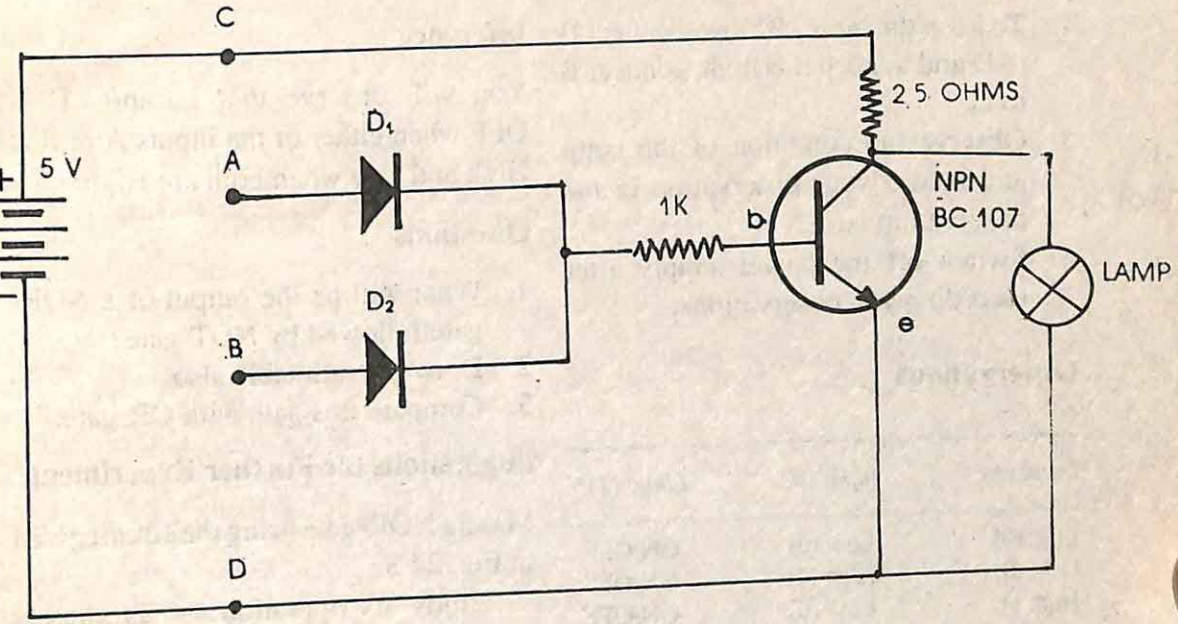


Fig 24.2

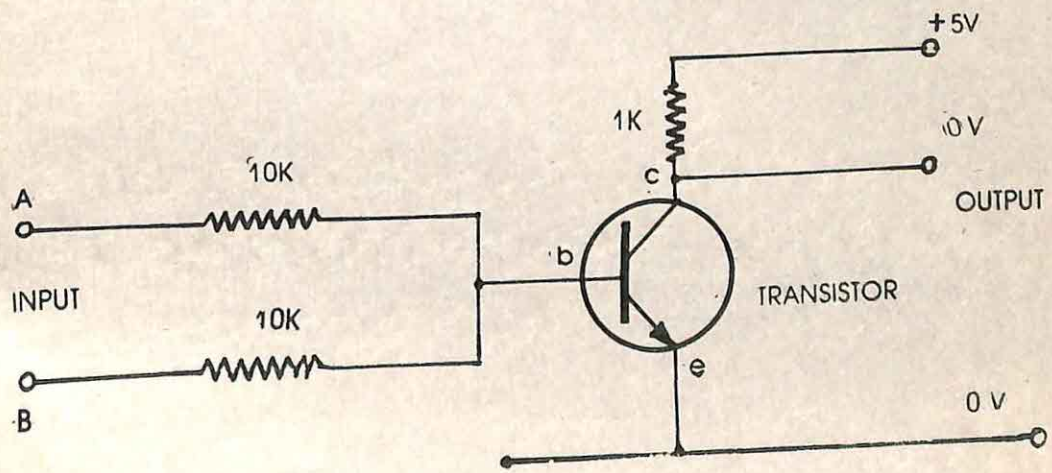


Fig. 24.3

4. To keep the input 'B' low, connect B to D and to keep this high, connect B to C.
5. Observe the condition of the lamp and record your observation in the observation table.
6. Switch off the power supply after recording you observations.

Observations

Input (A)	*Input (B)*	*Lamp (Y)**
Low (0)	Low (0)	ON/OFF
Low (0)	High (1)	ON/OFF
High (1)	Low (0)	ON/OFF
High (1)	High (1)	ON/OFF

*Strike off whichever is not applicable.

Inference

You will observe that Lamp/LED is OFF when either of the inputs A & B is High and also when both are High.

Questions

1. What will be the output of a NOR-gate followed by NOT-gate?
2. Draw its truth table also.
3. Compare this gate with OR-gate.

Suggestions for Further Experiment

Make a NOR-gate using the circuit given in Fig. 24.3.

Study its operation by alternately connecting input points A and B to high (1) and low (0) state as done on your experiment.

PRACTICAL PROJECTS

25. Construction of a Dual-power Supply Unit

Objective

You might have observed that you have to derive d.c. voltages for most of your experiment from power supply units readily available in your lab.

You have conducted experiments of half-wave and full-wave rectifiers, where you get either a positive or negative d.c. voltage at the output, depending on the direction in which you connect the rectifier diodes. In all these cases, electrolytic capacitors have been used to suppress the ripples. You have also conducted an experiment to regulate output d.c. voltage by using a zener diode.

It is however often necessary to derive both positive and negative voltages simultaneously when using operational amplifiers or such other circuit components. This project of constructing a dual power supply unit giving + 15V and −15V. will make you more familiar with electronic regulation where the output voltage is held constant within limits and ripple voltages are very low.

Circuit Diagram

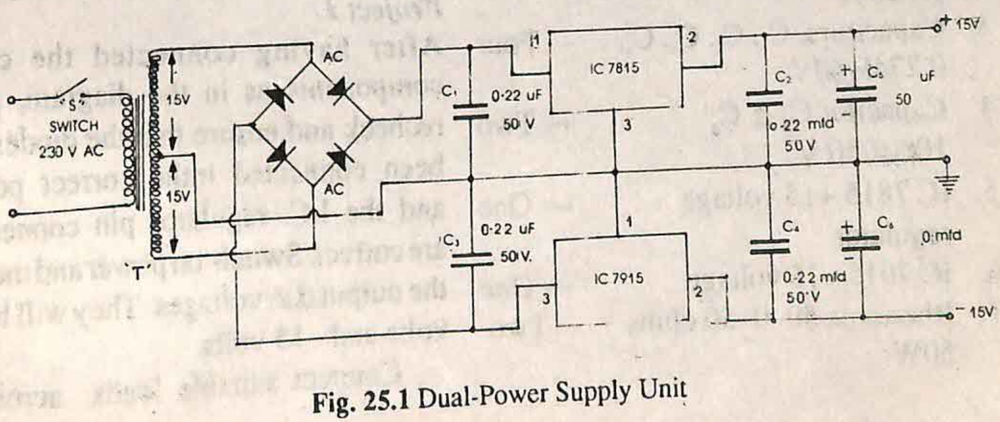


Fig. 25.1 Dual-Power Supply Unit

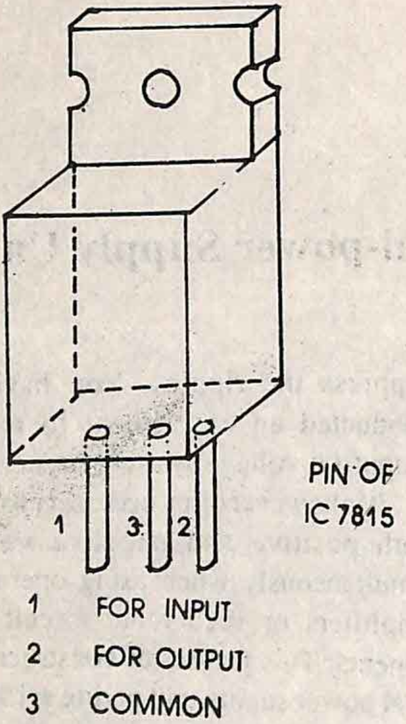


Fig 25.2

Materials Required

1. Step down power transformer — One
2. Semiconductor diode IN 4001 — Four
3. Capacitors, C_1, C_2, C_3, C_4, 0.224F/50V — Four
4. Capacitor C_5 & C_6 100µf/50 V — Two
5. IC 7815 +15 voltage regulator — One
6. IC 7915 –15 voltage — One
7. Rheostats 50–0–50 ohms 50W —Two

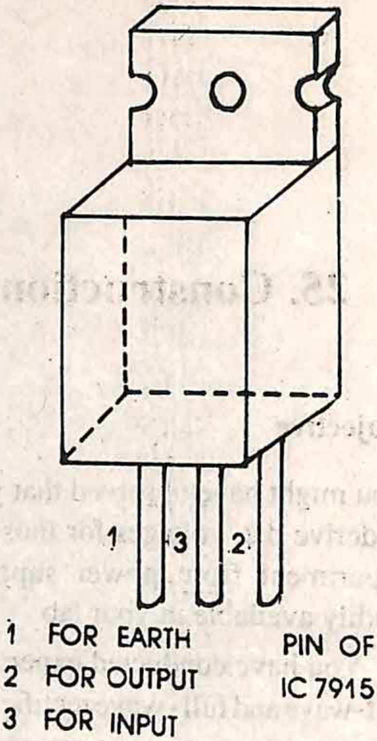


Fig 25.3

Circuit Symbol

IC 7815

Pin No. 1 input
2 output
3 common

IC 7915

pin no. 1 common
2 output
3 input

Project 1

After having connected the circuit components as in the diagram, please recheck and ensure that the diodes have been connected inthe correct polarity and the I.C. regulator pin connections are correct. Switch-on power and measure the output d.c. voltages. They will be +15 volts and –15 volts.

Connect suitable loads across the

positive and negative d.c. voltages. This may be done in two steps. First connect 30 ohms from a high-wattage Rheostat. The load current is 0.5 amps. Observe that the voltage is 15 volts.

Then connect 15 ohms and observe that the voltage still remains at 15 volts. This is due to the electronic regulation achieved due to the two electronic regulators.

Note

Heat sink for both ICs regulator 7815 & 7915 should be apart to each other.

With the help of an oscilloscope you will observe that the ripple at the output is extremely low.

26. Construction of Call Bell Using a IC-741

Objective

Operational amplifiers or op-amps have many uses, two of the most important being (i) as high gain voltage amplifiers of d.c. and a.c. (ii) as switches.

A typical op-amp like popular 741 contains about twenty transistors as well as resistors and small capacitors on the silicon chip.

A tone circuit using a 741 is very useful device and can be utilised as a door bell. The output of 741 is amplified by a complementary transistor pair (NPN and PNP) to give a loud tone in the speaker.

IC-741 is available in 14-pin dual-in-line, 8-pin-dual-in-line or in TO-style packages.

Pin configurations of 8-pin-dual-in line or TO-style package have been shown in the Fig. 26.1

The project of constructing an electronic call-bell circuit using op-amp 741 will make you familiar with the I.C.

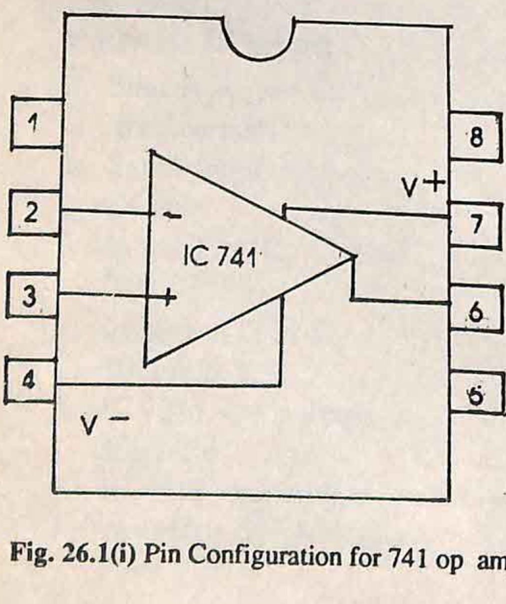


Fig. 26.1(i) Pin Configuration for 741 op amp

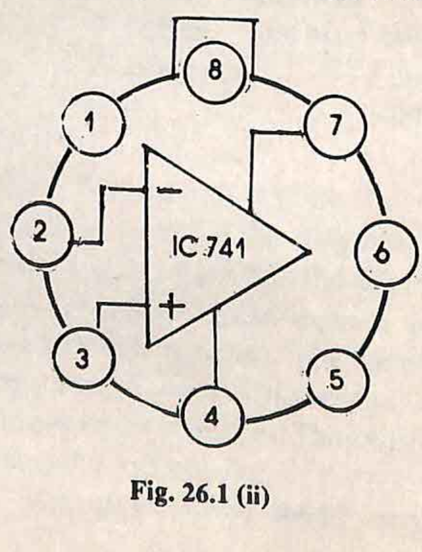


Fig. 26.1 (ii)

Circuit Diagram

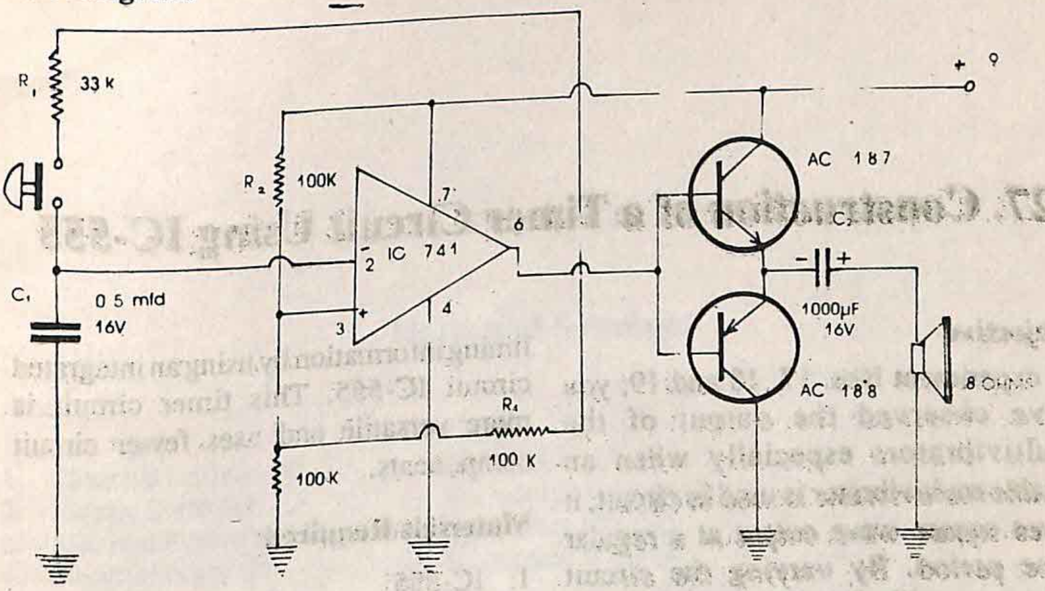


Fig. 26.2

chip and the proper method of connecting it to the circuit after taking some preliminary precaution .

Materials Required

1. IC – 741 with base 1No.
2. Resistances 100k - 1/4 3 No.
3. Resistance 33k-1/4 1 No.
4. Transistor AC 187 1 No.
5. Loudspeaker (8) 1 No.
6. Capacitor 100μF/16 volts 1 No.
7. Capacitor 0.5 μF 1 No.
8. Switch (push button type) 1 No.
9. Connecting wires etc.

Procedure

Before connecting the IC 741 in the circuit by plugging it into the socket ensure that the Pin number of the IC coincide with Pin numbers in the circuit board. Handle the IC with care so that Pins do not bent and break. After construction of the project, you may change the value of the resistor R_1, and note the change in tone produced by the loudspeaker.

27. Construction of a Timer Circuit Using IC-555

Objective

In experiment Nos. 17, 18 and 19, you have observed the output of the multivibrators especially when an astable multivibrator is used in circuit, it gives square wave output at a regular time period. By varying the circuit components like R and C we can vary the frequency of the square-wave output. These square-wave signals are later used as timing reference.

In this project, you will construct a timer circuit capable of giving variable timing information by using an integrated circuit IC-555. This timer circuit is more versatile and uses fewer circuit components.

Circuit Diagram

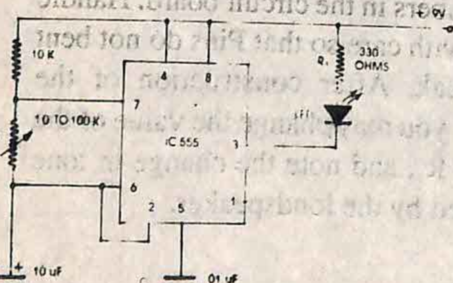


Fig. 27.1

Materials Required

1. IC-555
2. LED 1
3. Capacitors C_1 10 μF/12 V 1
4. C_2 0.01μF/50V 1
5. Resistor R_1 10K 1
6. R_2O-100K potentiometer 1
7. R_3-330Ω 1
8. Power supply unit 9V 1
9. Connecting wires etc.

Observation

When the circuit is energised, the LED goes ON/OFF at a regular frequency. Vary the potentiometer R_2 and frequency of the glow will change. The time period is given by the formulae.

$$T = 0.69\ (R_1 + R_2)\ C_1$$

ANNEXURE I

Units and Symbols

1. Electrical Potential (V) -- Measured in volts (V)
2. Electric Current (I) — Measured in Amperes (Amps)
3. Electrical Resistance (R) — Measured in ohms (Ω)
4. Electrical Power (P) — Measured in Watts (W)

Submultiple of Units

Milli volts	1000	mv	= 1V
Micro volts	1000	μv	= 1 mv
Milli amps	1000	ma	= 1 A
Micro amp	1000	μA	= 1 mA
	1000000	μA	= 1A
Milli watts	1000	mw	= 1W
Kilohms		1 K ohms	= $1000\Omega = 10^3\,\Omega$
		1 M ohms (mega)	= $1000K\Omega = 10^6\,\Omega$

ANNEXURE II

Resistors

Fixed Resistors

Basically resistors are used to limit the current in an electric circuit. When choosing a resistor three factors should be considered.

(a) The tolerance : A resistor with a stated value called nominal value. A fixed resistor of 100 ohm and a tolerance of ± 10% could have any value between 90 and 110 ohm 5% is normally good enough for most applications.

(b) The power rating of : ..... This is the maximum power which can be developed in a resistor without damage due to heating for most electronic circuits 0.25 or .5 W ratings are sufficient.

(c) The stability : this is the ability to keep the same value of a resistor with change in temperature when run for a long time.

For most circuits the cheapest type of resitors are of 0.25W carbon composition resistor of 5% tolerance is suitable. The 0.5 W or 0.6 W metal film resistor is more expensive but has higher power rating, smaller tolerance (percent) and much better heat stability. It is used in expensive equipment High-power resistors include

— wire would (3w, 7w, 10w usually for resistors more than a few ohms and fraction of ohms)

— metal film (3w, 7w, 10w usually for resistance more than a few ohms up to about 4 .7k)

— wire would ceramic (4w to 17w or more wide range of values, expensive).

Resistance colour codes

Colours and digits

Black =0
Brown =1
Red = 2
Orange =3
Yellow =4
Green =5
Blue =6
Violet =7
Grey =8
White =9

Resistance is shown by 3 bands

Band 1

Band 1)

Band 2) first two digits

Band 3) The number of 3 ..... following (multiplier powers of 10)

Tolerance band

No band = ± 20%

Silver = ± 10%

Gold = ± 5%

Red = ± 2%

Brown = ± 1%

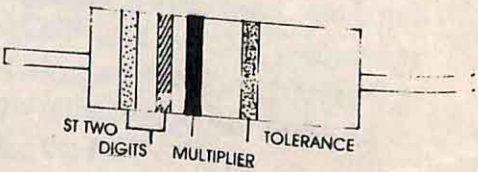


Fig. 1

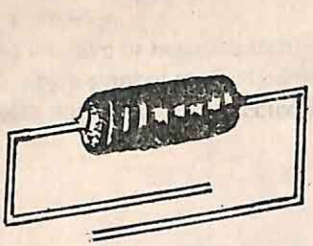


Fig. 2

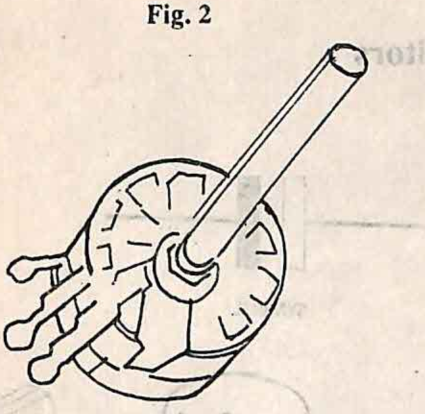


Fig. 3

Variable Resistor

A variable resistor is also called a potentiometer and is used for volume or other control in electronic equipment.

The minimum value of potentiometer is usually 100 ohm.

They are available in range of values such as 100, 220, 470, 1K, 22K, ohm and so on.

Potentiometers are used when the resistance has to be adjusted often or when higher power rating is required. Potentiometers are either rotary or sliding. They may be mounted on the circuit board or on a panel.

Carbon track potentiometers are cheaper but may wear out early tolerance is usually 20%. Cement track potention meters are expensive but give smooth control of resistance and tolerance is usually 10%. There are two types of (i) linear (ii) logarithmatic potentiometers. Linear tracks give an equal change of resistance for an equal amount of turn. Logarithmic potentiometers give large changes of resistance for a given amount of turn as one end of the track logarithmic potentiometers are used mainly for volume controls.

Carbon potentiometers are rated as up to 0.25w wire wound or cement potentiometers are used for higher power.

Annexure III

Capacitors

When choosing a capacitor, the main points to consider about in the value of the capacitor and its working voltage.

The unit of capacitance farad F is a large unit, therefore, for practical purposes sub-multiples of the farad (F) are used

Microfarad (µF)	= 10^{-6}F
nanofarad (nF)	= 1nf = 10^{-9}F
picofarad (pF)	= 1pf = 10^{-12}F

In circuit diagrams, values of capacitance in microfarads, nanofarads and picofarads are represented by µ, n and p. Capacitors of value of 1µf and over is considered as high capacitance and in normal conditions electrolytic capacitors are used in this range.

Normally polyester or ceramic capacitors are used in the ranges of 0.01 µf to 1 µF. For low capacitance (2.2 pf to 1000 PF) normally capacitors or ceramic capacitors are used.

The maximum voltage (d.c. or peak A.C.) which a capacitor can withstand before the dielectric breaks down is the working voltage of a capacitor. A capacitor will be damaged if the p.d. across it is greater than the working voltage. For 1000V and above propylene capacitors are used. The working voltage of polyester and ceramic capacitor is generally between 100 v and

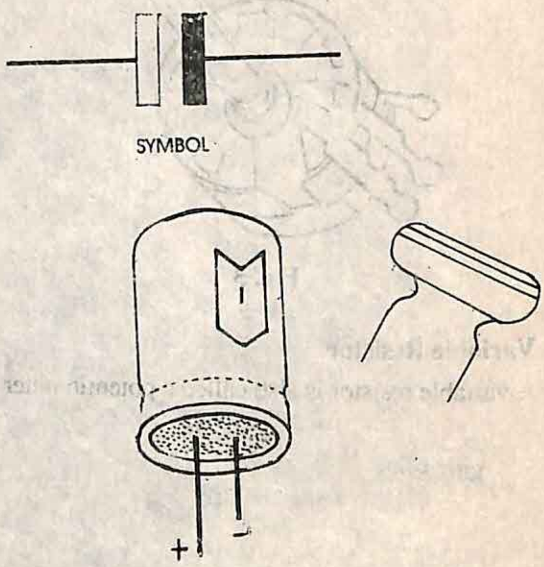


Fig. 1

600 v, which means that they can be used in almost any circuit without trouble. Electrolytic capacitors are made in a number of working voltage ranges. Generally the higher the working voltage, the larger the capacitor and more expansive. Electrolytic capacitors are suitable in circuits whose working voltage is 10v or 25 v.

Electrolytic capacitors are polar capacitors,

which means that these have a positive and negative polarity.

The positive or negative terminal is usually indicated by a symbol marked on the case. Polar capacitors must not be connected in the wrong way. This should not be used in a circuit in which the polarity of the voltage may sometimes be reversed. Polarized capacitors must be connected so that there is d.c. through them in the correct direction.

APPENDIX

List of Contributors

1. Shri N. N. Mohanty
   Formerly Director Staff Training
   Institute AIR and Doordarshan
   New Delhi

2. Shri M.M. Kaushik
   Birla Senior Secondary School
   Kamla Nagar
   Delhi

3. Srimati Ujjwala Gautam
   Electronics Teacher

4. Dr. Vijendra Sharma
   A.R.S.D. College
   Delhi University
   Delhi

5. Srimati Himali Pant
   Govt. Girls Senior
   Secondary School
   Lajpat Nagar
   New Delhi

6. Shri Raksh Pal Singh
   Workshop Department
   NCERT
   New Delhi

7. Shri N. P. Bhattacharya
   Department of Vocationalization
   of Education
   NCERT
   New Delhi

8. Shri P.K. Mohanty
   Reader
   Regional College of Education
   Bhubaneswar
   Orissa